DOCUMENT RESUME

ED 097 512 CE 002 280

AUTHOR	Shipp, Robert E.
TITLE	Developing Job Sheets and Related Aids for Individualized Instruction in the Machine Shop.
INSTITUTION	Tennessee Univ., Knoxville. Occupational Research and Development Coordinating Unit.
NOTE	422p.; Part of page 58, section 4, is illegible
EDRS PRICE	MF-$0.75 HC-$19.80 PLUS POSTAGE
DESCRIPTORS	*Hand Tools; Individual Activities; *Individualized Instruction; *Industrial Arts; *Instructional Materials; *Machine Tools; School Shops
IDENTIFIERS	*Machine Shop

ABSTRACT

The document is divided according to the four different kinds of sheets presented. The first section contains assignment sheets on 28 machine shop topics and supplementary transparencies. Included in the information presented is the title of the sheet, the unit and occupation to which it applies, the objective, reference for information, directions, and a list of questions. The second section contains sheets on related information, usually dealing with tools or procedures for their use. Twenty-five topics are covered with title, unit, occupation, objective, references, an introduction, and information specified for each sheet. Operation sheets are in section 3; seven operations are included specifying title, unit, occupation, objective, introduction, reference, and procedure for each. The final section contains job sheets for 25 tools used in machine shop. In this section, the title, unit, occupation, objective, information, specifications, materials, tools and equipment, and procedures are outlined. A cross-index is provided, keyed to the job sheets, for relating the job, assignment, operation, and information sheets to each other. (AG)

FT CB1

DEVELOPING JOB SHEETS AND RELATED AIDS FOR INDIVIDUALIZED INSTRUCTION IN THE MACHINE SHOP

A Mini-Grant Research Project

Presented to

The Tennessee Research Coordinating Unit

Robert E. Shipp

Machine Shop Instructor

Smyrna High School

BEST COPY AVAILABLE

ACKNOWLEDGMENTS

Appreciations are expressed to the following for their contributions to the development and duplication of the High School Job Sheets and Related Aids.

--Staff of the Tennessee Research Coordinating Unit who funded and assisted with the research project.

--Mr. Raymond Manley, Supervisor of Vocational Education, Rutherford County Board of Education, Murfreesboro, Tennessee, who reviewed the proposal and the developed materials.

--Mr. Calvin Duggin, Associate Professor Machine Shop, Middle Tennessee State University, Murfreesboro, Tennessee, who critiqued the materials developed.

--Mrs. Kathey Meadow for typing all of the job sheets and supplementary aids.

--Mr. Harry Gerhart for drafting of the prints for the job sheets.

--Mrs. Sue Messick, VOE Instructor at Smyrna High School, and the VOE students for duplication of materials.

In addition to the above, special thanks are extended to the T and I students and supervisors who helped in the development of the project.

Robert E. Shipp
Project Director
Smyrna High School

BEST COPY AVAILABLE

TABLE OF CONTENTS

Assignment Sheets

BEST COPY AVAILABLE

Related Information

Operation Sheets

BEST COPY AVAILABLE

Job Sheets

C R O S S   I N D E X

The index below should serve as a guide to the instructor in using and relating the job, assignment, operation and information sheets. All related aid sheets (assignment, operation, and information) are keyed to the job sheets.

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
Drill Drift	Shop Safety, Hand tools, Hacksaws and sawing, Files and filing and measuring and layout tools.		Safety in the shop, Finishing, Sp eds and feeds for drilling.
Hammer Head	Shop safety, Hand tools, Hacksaws and sawing, Files and filing and measuring and layout tools		Safety in the Shop, Finishing, Speeds and feeds for drilling.
Step [illegible]	Shop Safety, Measuring and layout, power saws and sawing, cutting tools, lathe and lathe operations, To find R.P.M. and cutting speed of a lathe	To Grind a general purpose turning tool, To knurl work in the lathe	Safety in the Shop, Finishing, How to read a micrometer, Single point cutting tools methods of holding work on a lathe
Hammer Handle	Shop Safety, Power saws and sawing, measuring and layout tools, cutting tools, Lathe and lathe operation, Machine taper, Drills and Drilling Processer,	To Grind a general purpose Turning tool and to knurl work in the lathe,	Safety in the shop, Finishing, How to read a micrometer, Single point cutting tools, methods of holding work on a lathe, Types of tapers and Taper formula, speeds and feeds for reaming.

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
Plumb Bob	Shop Safety, Power saws and sawing, measuring and layout tools, cutting tools, Lathe and lathe operations, and To find the R.P.M. and cutting speed of a lathe,	To grind a general purpose turning tool and To grind a drill	Safety in the Shop, How to read a micrometer, Drills and drill sizes, Single point cutting tools, methods of holding work in a lathe and Types of tapers, Taper formulas.
V-Block	Shop Safety, Power saws and sawing, Measuring and layout tools, milling machines and milling, Shaper and shaper operations, To find the R.P.M. and C. S. of a milling machine, Heat treatment of steel and surface grinding and grinding operations	To harden and temper carbon tool steel	Safety in the Shop, How to read a micrometer, Operation of shaper and manufacture of steel
V-Block Clamp	Shop Safety, Measuring and layout tools, Drills and drilling Processes, Milling machines and milling, To find the R. P. M. and C. S. of a mill, types of screw threads, Power saws and sawing, Files and filing, Cutting tools, Lathe and lathe operations, To find the R.P.M. and C.S. of a lathe.	To grind a general purpose turning tool, To grind a drill and Thread cutting.	Safety in the shop. How to read a micrometer, Drills and drill sizes, Speeds and feeds for drilling, single point cutting tools, operation of a shaper and Screw thread terms.

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
C- Clamp	Shop Safety, Measuring and layout tools, Drills and drilling processes, milling machines and milling, Shaper and Shaper operations, Surface grinding and grinding operations, Files and filing, cutting tools, Lathe and lathe operations and Types of screw threads.	To Grind a general purpose turning tool To grind a drill and Thread cutting	Safety in the shop, How to read a micrometer, Drills and drill sizes, Speeds and feeds for drilling, Single point cutting tools, operations of a shaper and Screw thread terms.
60° Lathe Center	Shop Safety, Power Saws and sawing, measuring and layout tools, cutting tools, Lathe and lathe operations, To find the R. P. M. and C. S. of a lathe and machine tapers.	To grind a general purpose turning tool.	Safety in the shop, How to read a micrometer, Single point cutting tools, Types of tapers.
Machinist's Clamp	Shop safety, Power saws and sawing, Files and filing, measuring and layout tools, Drills and drilling processer, cutting tools, Lathe and lathe operations, R. P. M. and C. S. of a lathe, Types of screw threads, Shaper and Shaper operations.	To Grind a drill, To grind a general purpose turning tool and thread cutting.	Safety in the Shop, Finishing, How to read a micrometer, Drills and drill sizes, speeds and feeds for drilling, Single point cutting tools, Screw thread terms, Screw thread formulas, operation of shaper.
Ball-Peen Hammer	Shop Safety, Power saws and sawing, Lathe and lathe operations, cutting tools, measuring and layout tools, Drills and Drilling Processes and Milling machine and milling	To grind a general purpose turning tool, To grind a drill and To harden and temper carbon tool steel	Safety in the shop, Finishing, How to read a micrometer, Speeds and feeds for drilling, Single point cutting tools, Manufacture of steel.

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
Hand Center Punch	Shop Safety, Powersaws and sawing, measuring and layout tools, cutting tools, Lathes and lathe operations, machine tapers, Drills and drilling.	To grind a general purpose turning tool, To grind a drill	Safety in the shop, Finishing, How to read a micrometer, Single point cutting tools, Taper formulas.
Parallels	Shop Safety, Power saws and sawing, measuring and layout tools, Drills and drilling processes, milling machine and milling, Shaper and shaper operations, Heat treatment of steel and Surface grinding and grinding operations.	To grind a drill	Safety in the Shop, How to read a micrometer, Speeds and feeds for drilling and operations of a shaper.
Hold-Downs	Shop safety, Power saws and sawing, measuring and layout tools, milling machine and milling, Shaper and shaper operations, Heat treatment of steel and Surface grinding and grinding operations.		Safety in the shop, operation of a shaper, and Manufacture of steel.
Bench Block	Shop safety, Power saws and sawing, measuring and layout tools, Drills and drilling processes, Lathe and lathe operations, Heat treatment of steel and Surface grinding and grinding operations.	To grind a general purpose turning tool, To grind a drill and to harden and temper carbon tool steel.	Safety in the shop, How to read a micrometer, Speeds and feeds for drilling, Speeds and feeds for reaming, Single point cutting tools, Boring on the lathe,

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
"T" Handle Tap Wrench	Shop safety, power saws and sawing, measuring and layout tools. Drills and drilling processes, cutting tools. Lathe and lathe operations, machine tapers, Types of screw threads, milling machine and milling and Heat treatment of steel	To grind a general purpose turning tool, To knurl work in the lathe, To grind a drill, Thread cutting and Internal threading on the lathe.	Safety in the shop, How to read a micrometer, Speeds and feeds for drilling. Boring on the lathe, Taper formula
Adjustable Tap Wrench	Shop safety, power saws and sawing, Files and filing, measuring and layout tools, Drills and drilling processes, cutting tools, Lathes and lathe operations, Types of screw threads, Milling machines and milling and Heat treatment of steel.	To grind a general purpose turning tool, To knurl work in the lathe, To grind a drill, Thread cutting Spring winding on the lathe and to harden and temper carbon tool steel.	Safety in the shop, Finishing, How to read a micrometer, Speeds and feeds for drilling, Single point cutting tools, Screw thread formulas,
Die Stock	Shop Safety, Power saws and sawing, Lathes and lathe operations, measuring and layout tools, Drills and drilling processes, machine tapers, Types of screw threads, milling machine and milling	To grind a general purpose turning tool, To knurl in the lathe, To grind a drill, and Thread cutting	Safety in the Shop, How to read a micrometer, Speeds and feeds for drilling, Single point cutting tools, Taper formulas, Screw thread formulas.
Machinist's Jack Screw	Shop Safety, Power Saws and sawing, Hacksaws and sawing, measuring and layout tools, Lathes and lathe operations, Drills and drilling processes and cutting tools.	To grind a general purpose turning tool, To grind a drill, Threading and Internal threading on the lathe.	Safety in the shop, How to read a micrometer, Speeds and feeds for drilling, Single point cutting tools, Boring on the lathe and screw thread formulas

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
Surface Gage	Shop safety, Power saws and sawing, Files and filing, measuring and layout tools, Drills and drilling processes, cutting tools, Lathe and lathe operations, milling machine and milling.	To grind a general purpose turning tool, To grind a drill and Thread cutting.	Safety in the shop, How to read a micrometer, Speeds and feeds for drilling. Single point cutting tools, Boring on the lathe,
Machinist Vise	Shop safety, Power saws and sawing, Files and filing, measuring and layout tools, Drills and drilling processes, Lathes and lathe processes, Types of screw threads, milling machines and milling, Shaper and shaper operations Surface grinding and grinding operations and Heat treatment of steel	To grind a general purpose turning tool, To mount work in the lathe, To grind a drill, Thread cutting and to harden and temper carbon tool steel	Safety in the shop, Finishing, How to read a micrometer, How to read an inside micrometer or depth micrometer, Speeds and feeds for drilling, Single point cutting tools, operation of a shaper.
[illegible] Sine Bar	Shop safety, Power saws and sawing, measuring and layout tools, Drills and drilling processes, lathes and lathe operations, Milling machines and milling, Surface grinding and grinding operations, Clyindrical grinding and grinding operations, Heat treatment of steel and surface finishes and measurements.	To grind a general purpose turning tool, To grind a drill and to harden and temper carbon tool steel	Safety in the Shop, Finishing, How to ream a micrometer, How to read an inside micrometer or a depth micrometer Speeds and feeds for drilling, Single point cutting tools.

BEST COPY AVAILABLE

JOB SHEETS	ASSIGNMENT SHEETS	OPERATION SHEETS	INFORMATION SHEETS
Hand Vise	Shop Safety, Power saws and sawing, Files and filing, measuring and layout tools, Drills and drilling processes, cutting tools, Lathe and lathe operations, Machine tapers, milling machines and milling operations, Abrasives and abrasive products and Heat treatment of steel	To Grind a general purpose turning tool, To knurl work in the lathe, To grind a drill, Thread cutting, To harden and temper carbon tool steel	Safety in the Shop, Finishing, How to read a micrometer, Speeds and feeds for drilling, Single point cutting tools, Taper formulas, Screw thread formulas, The rotary table and its use
Arbor Press	Shop Safety, Power saws and sawing, Files and filing, measuring and layout tools, Drills and drilling processes, Lathe and lathe operations, Milling machines and milling, Shaper and shaper operations and types of gears,	To grind a general purpose turning tool and to grind a drill.	Safety in the shop, Finishing, How to read a micrometer, How to an inside micrometer or depth micrometer, Speeds and feeds for drilling, Single point cutting tools, The rotary table and its use, Spur Gear terminology and spur gear formulas

TITLE: SHOP SAFETY

UNIT: SAFETY

OCCUPATION: MACHINIST

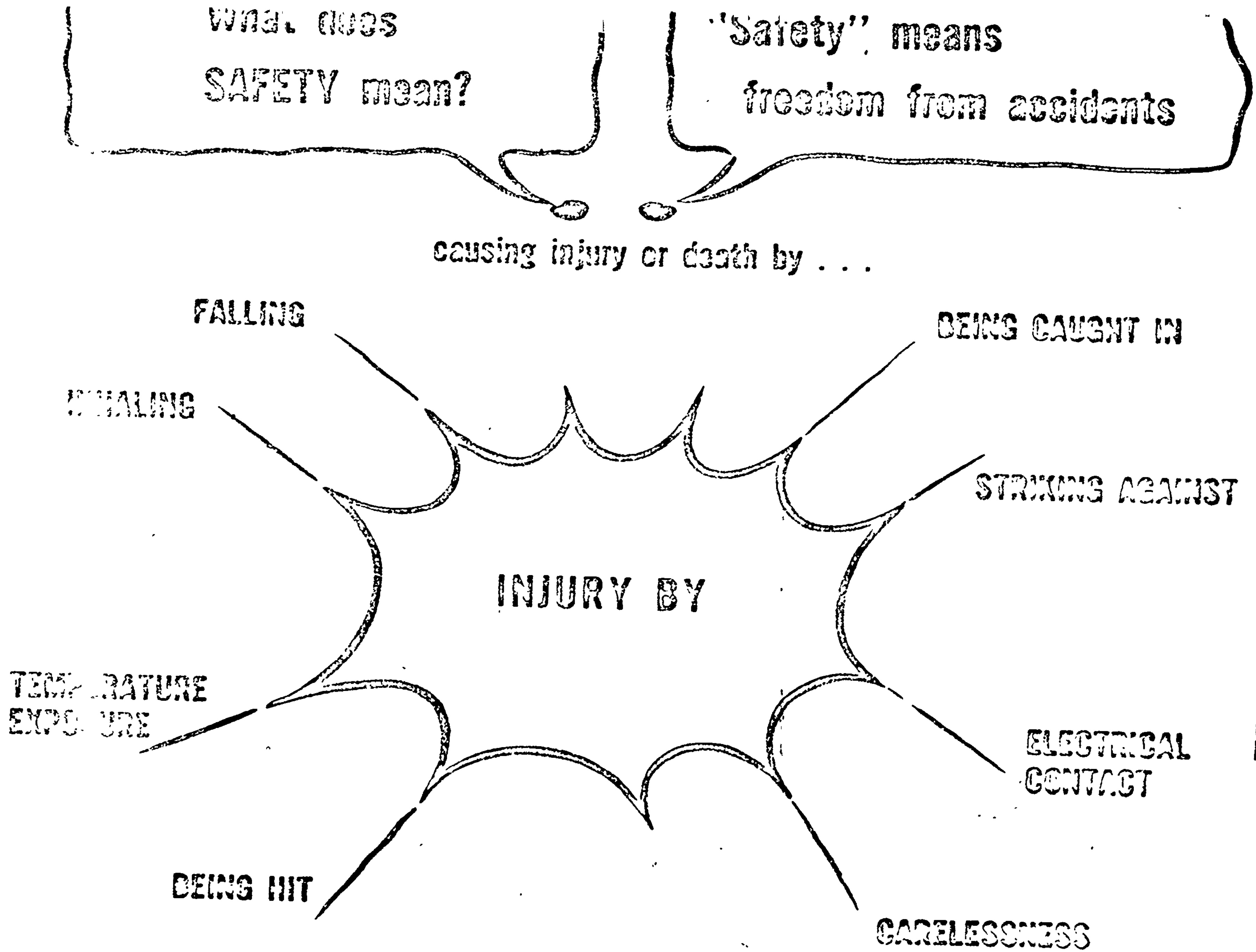
OBJECTIVE: To familiarize the student with common safety practices in the shop.

REFERENCE: Anderson - Tatro. Shop Theory New York; McGraw-Hill Book Co., Inc. Chapter 1, page 1-22

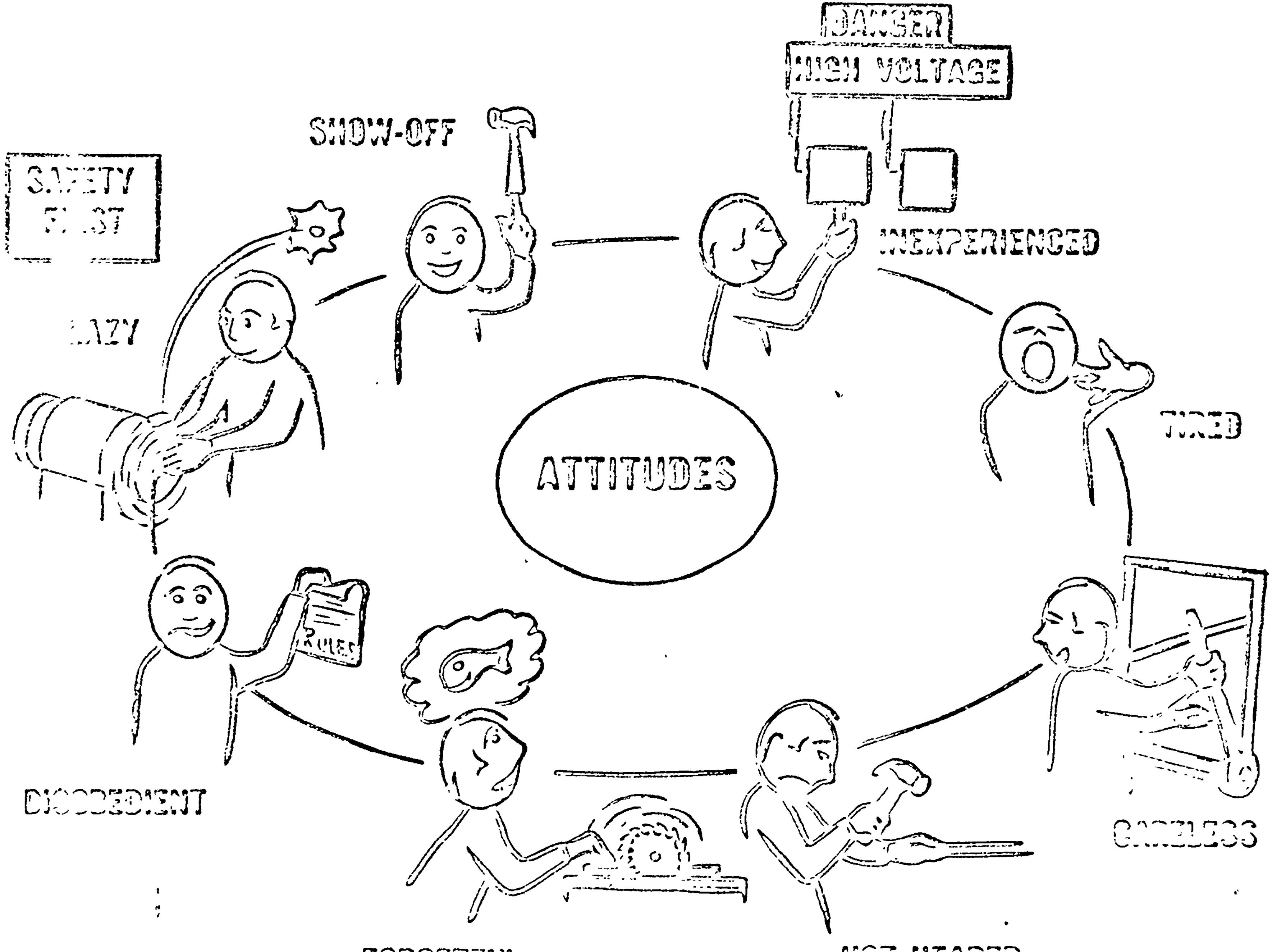
DIRECTIONS: Read the above reference and answer the following questions:

QUESTIONS:

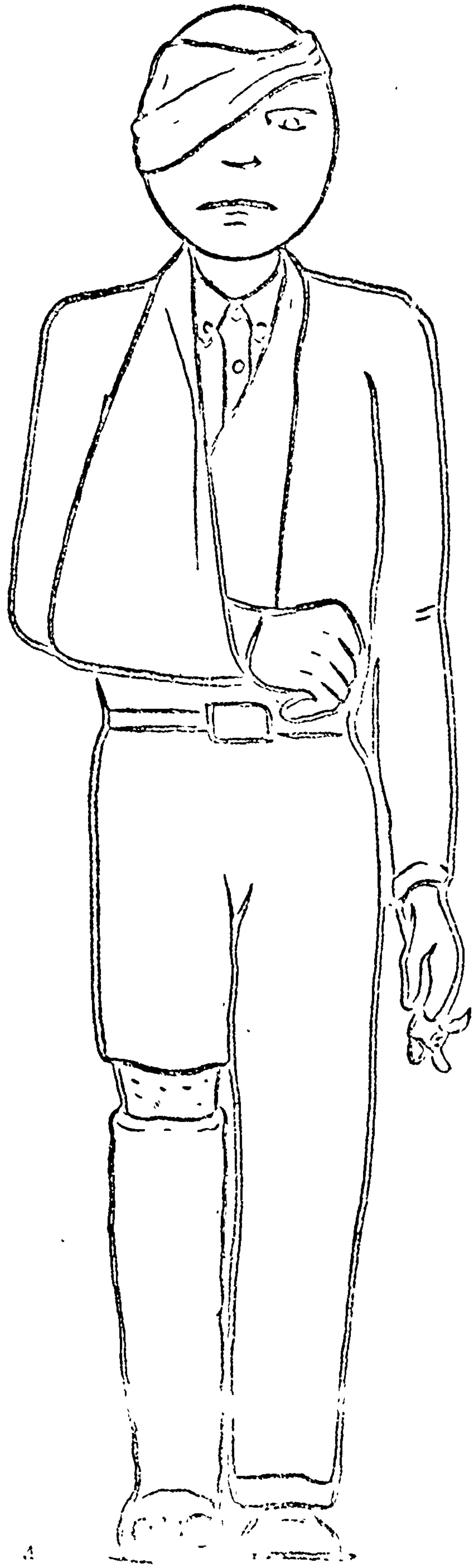
1. What is a safe dress for a machinist?
2. How should a long steel bar of stock be carried in the shop?
3. What is the safe way to lift a heavy object?
4. What is meant by good houskeeping in a machine shop?
5. Why is it dangerious to run a machine on which the guards have been removed?
6. What causes the greatest number of accidents to workers on the bench?
7. What are the rules that govern the safe use of wrenches?
8. What rules govern the practice of safe hacksawing?
9. How should work be held for drilling on the drill press?
10. What rules govern the safe operation of drill press?
11. What point of safety does the machinist stress when he says, "Never let go of the chuck key"?
12. How should milling cutters be handled?
13. List five (5) of the most important rules to follow that will assure the safe operation of a milling machine?
14. Why is it important that the operator of the shaping machine wear safety glasses?
15. List seven of the most important rules that govern the safe use of a grinding machine?


What does
SAFETY mean?
"Safety" means
freedom from accidents
causing injury or death by . . .
FALLING
BEING CAUGHT IN
HALING
STRIKING AGAINST
INJURY BY
TEM RATURE
EXPO URE
ELECTRICAL
CONTACT
BEING HIT
CARELESSNESS

STATE OF MIND – CAUSES ACCIDENTS



IF INJURED . . . where will you hurt?



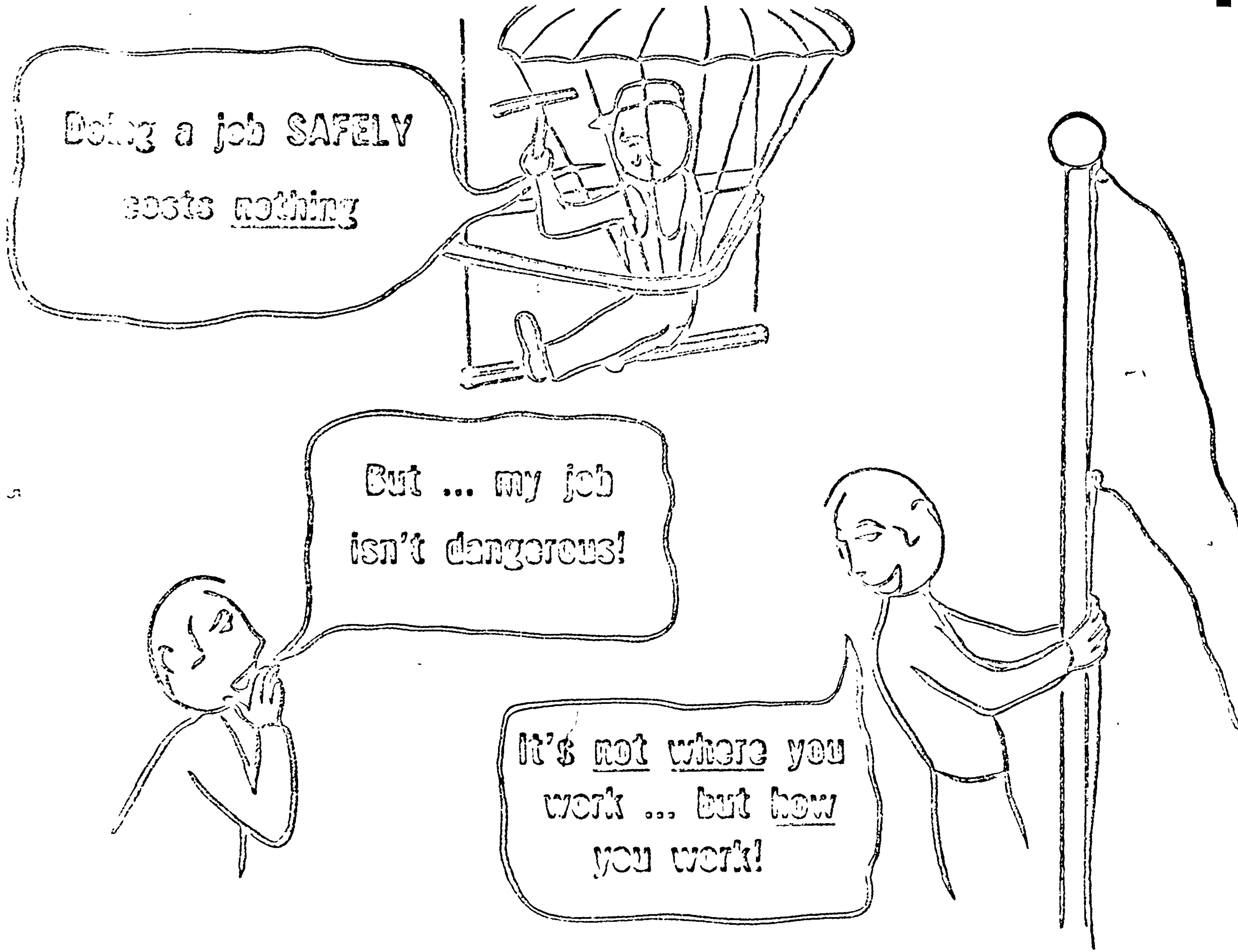
10% HEAD

25% BODY

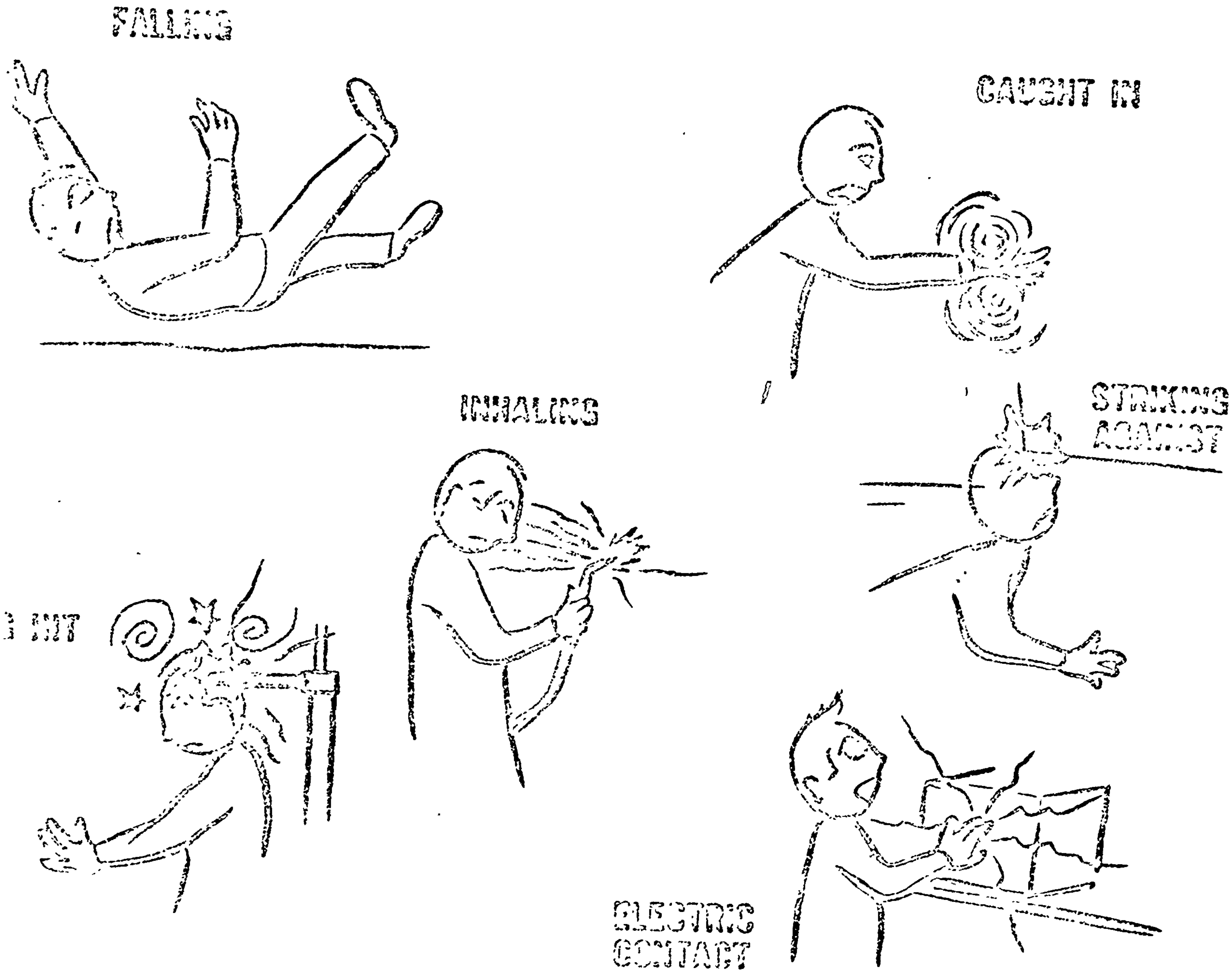
36% ARMS AND HANDS

12% LEGS

12% FEET


Doing a job SAFELY costs nothing
But ... my job isn't dangerous!
It's not where you work ... but how you work!

SO BEWARE OF . . .


FALLING
CAUGHT IN
INHALING
STRIKING
AGAINST
HIT
ELECTRIC
CONTACT

"LITTLE HURTS" LIKE

BLISTERS

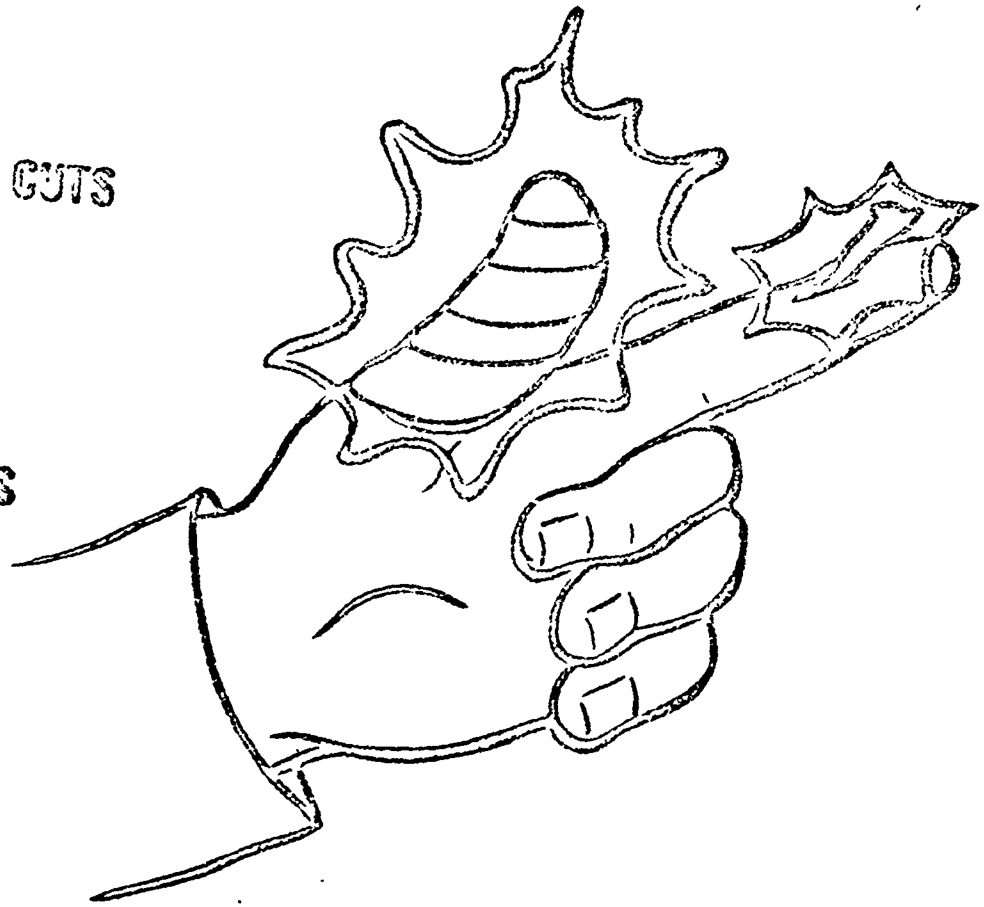
SPLINTERS

SMALL CUTS

SOMETHING IN EYE

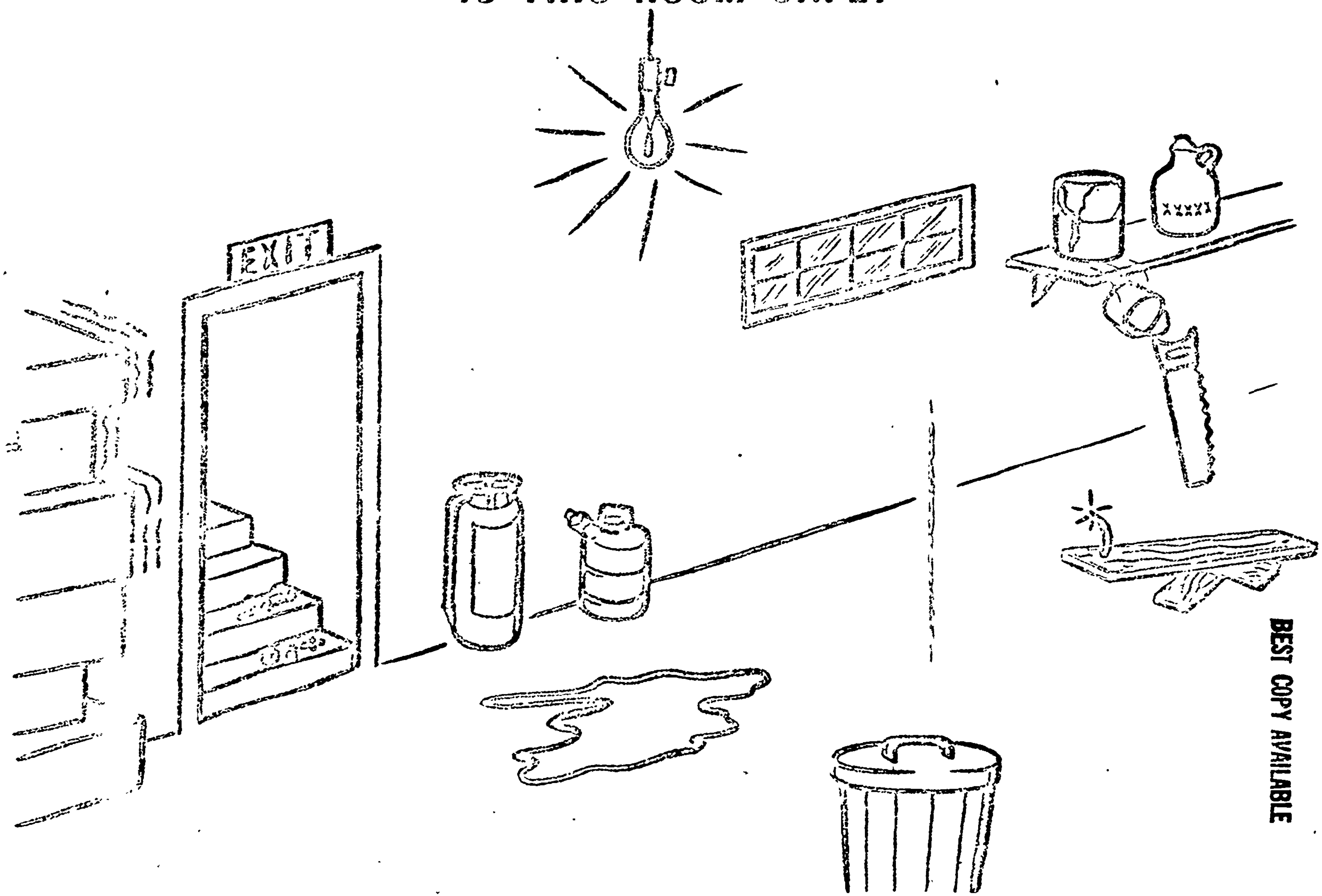
JAMMED FINGERS

BURNS



GET TREATMENT IMMEDIATELY

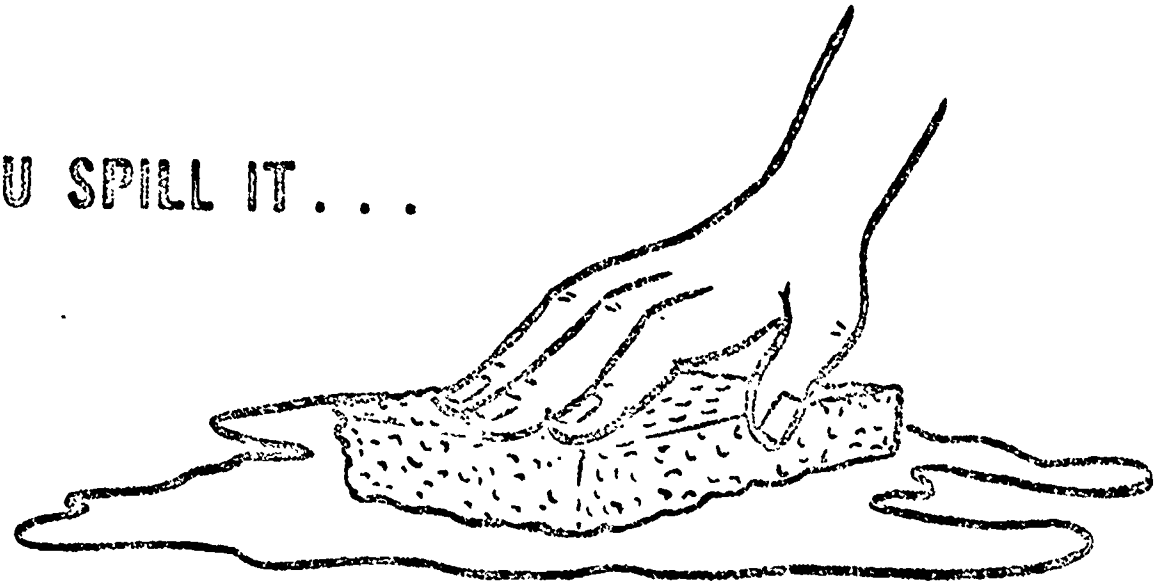
IS THIS ROOM SAFE?



HELP IMPROVE ROOM SAFETY

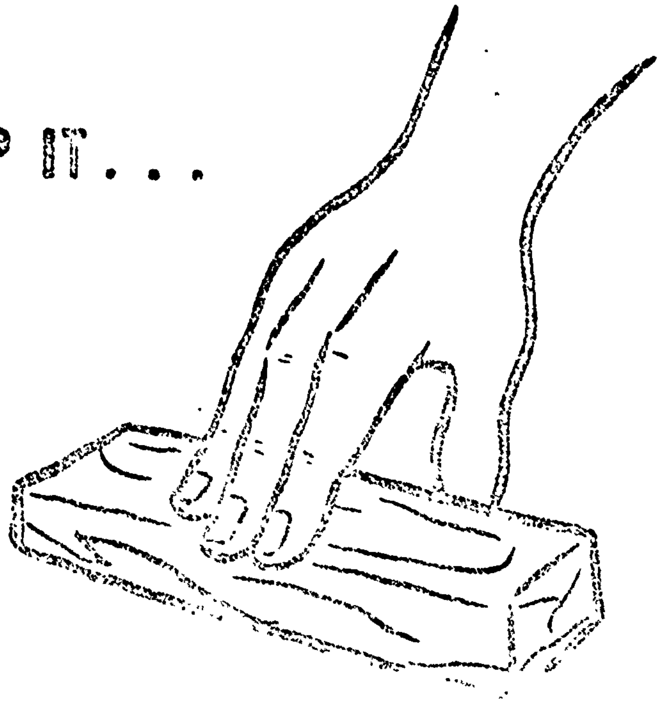
PREVENT A FALL. BEST COPY AVAILABLE

IF YOU SPILL IT...



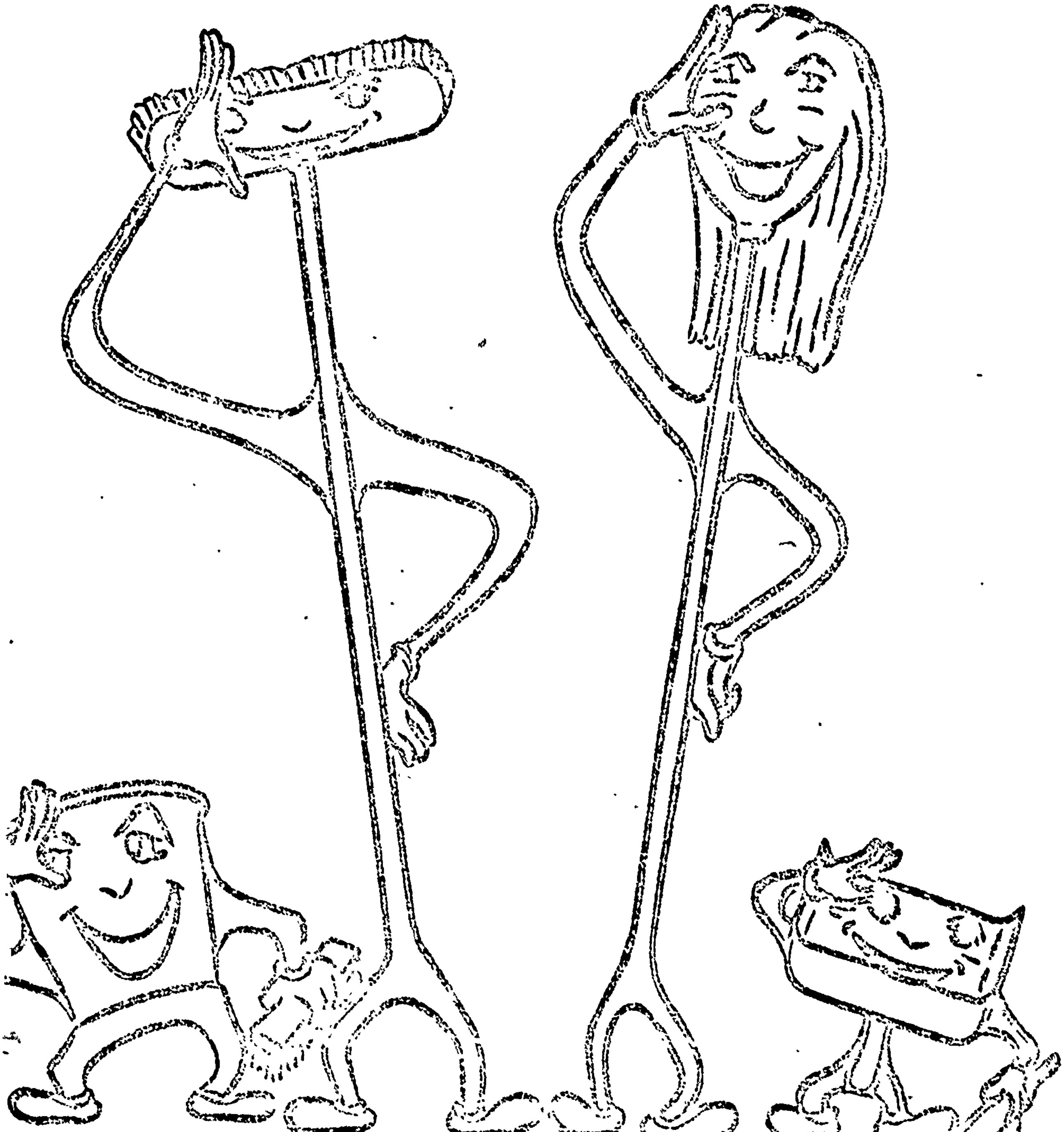
WIPE IT UP.

F YOU DROP IT...



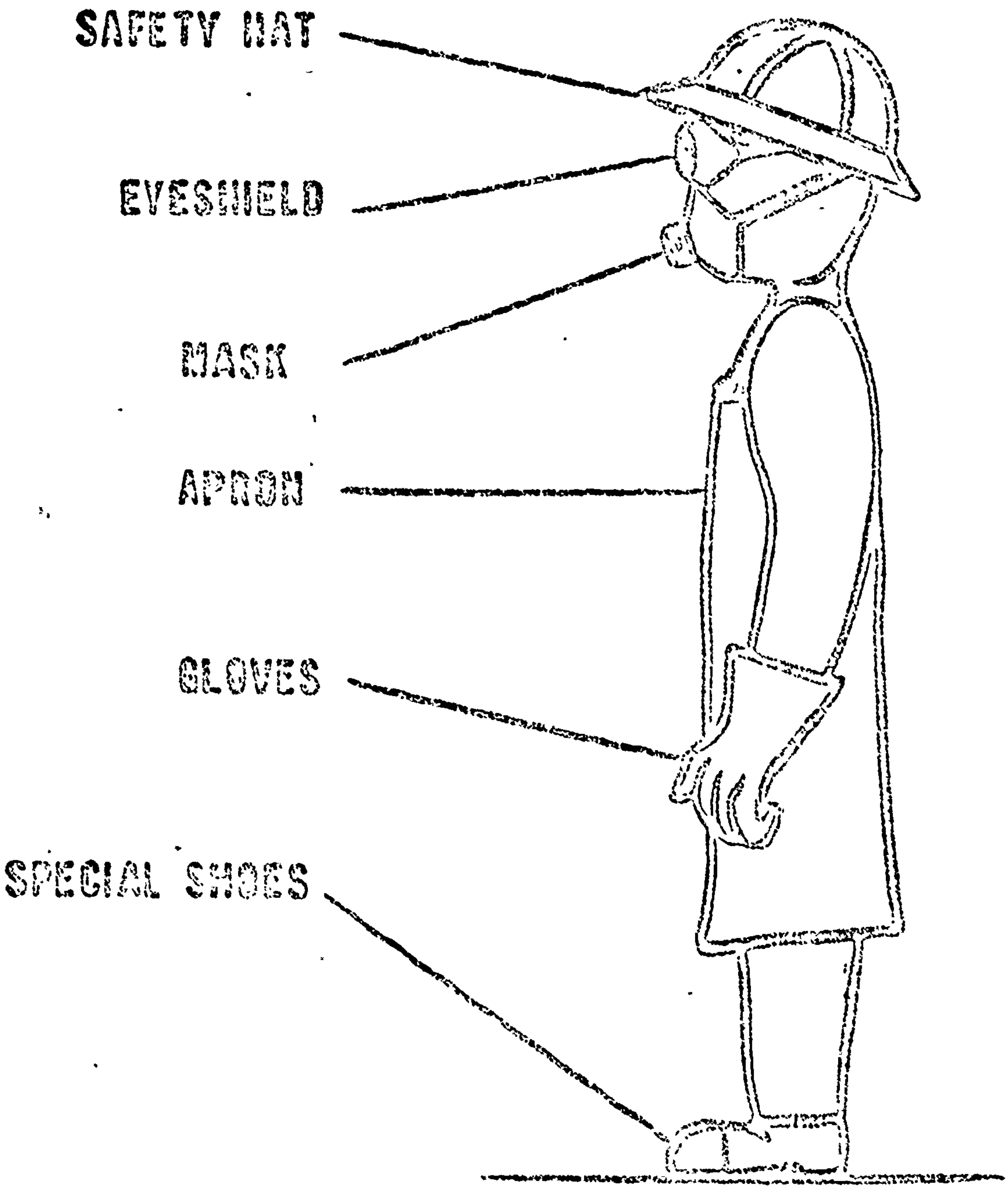
PICK IT UP.

TIME FOR CLEAN-UP

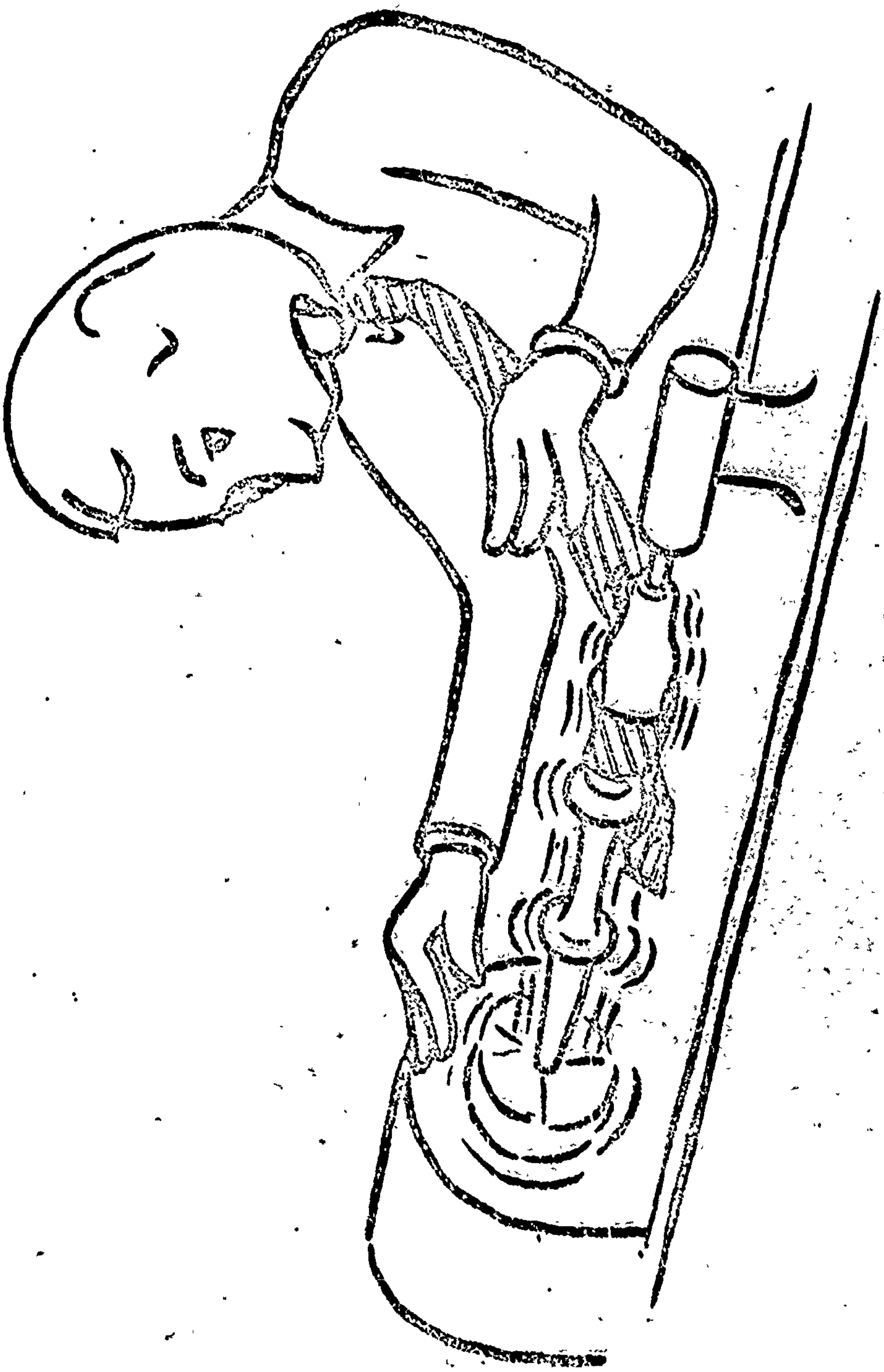


A CLEAN SHOP
IS A HEALTHY SHOP

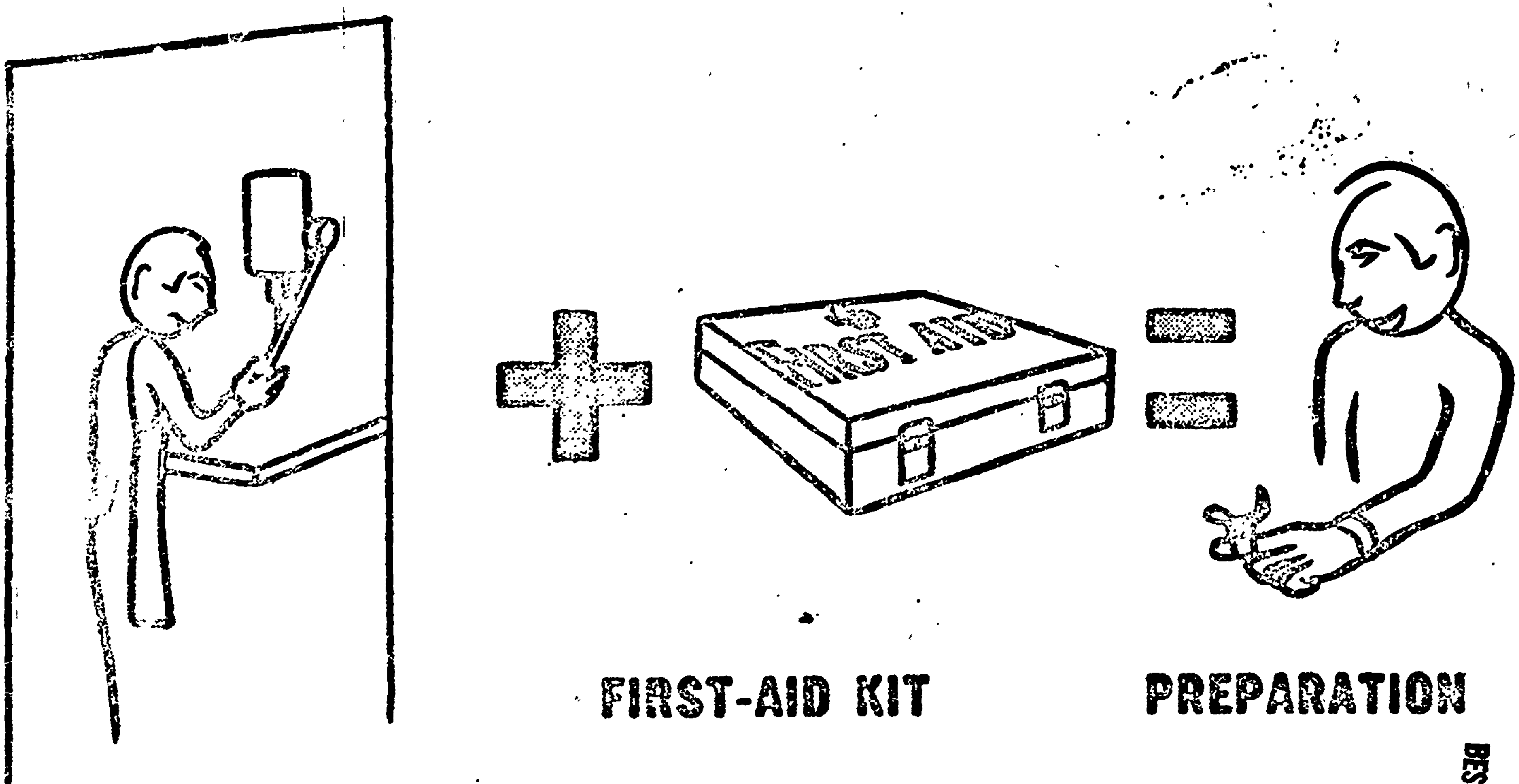
HOW IMPORTANT IS CLOTHING?
SOME JOBS REQUIRE SPECIAL CLOTHING



[illegible]
WEARING THE RIGHT PROTECTIVE CLOTHING



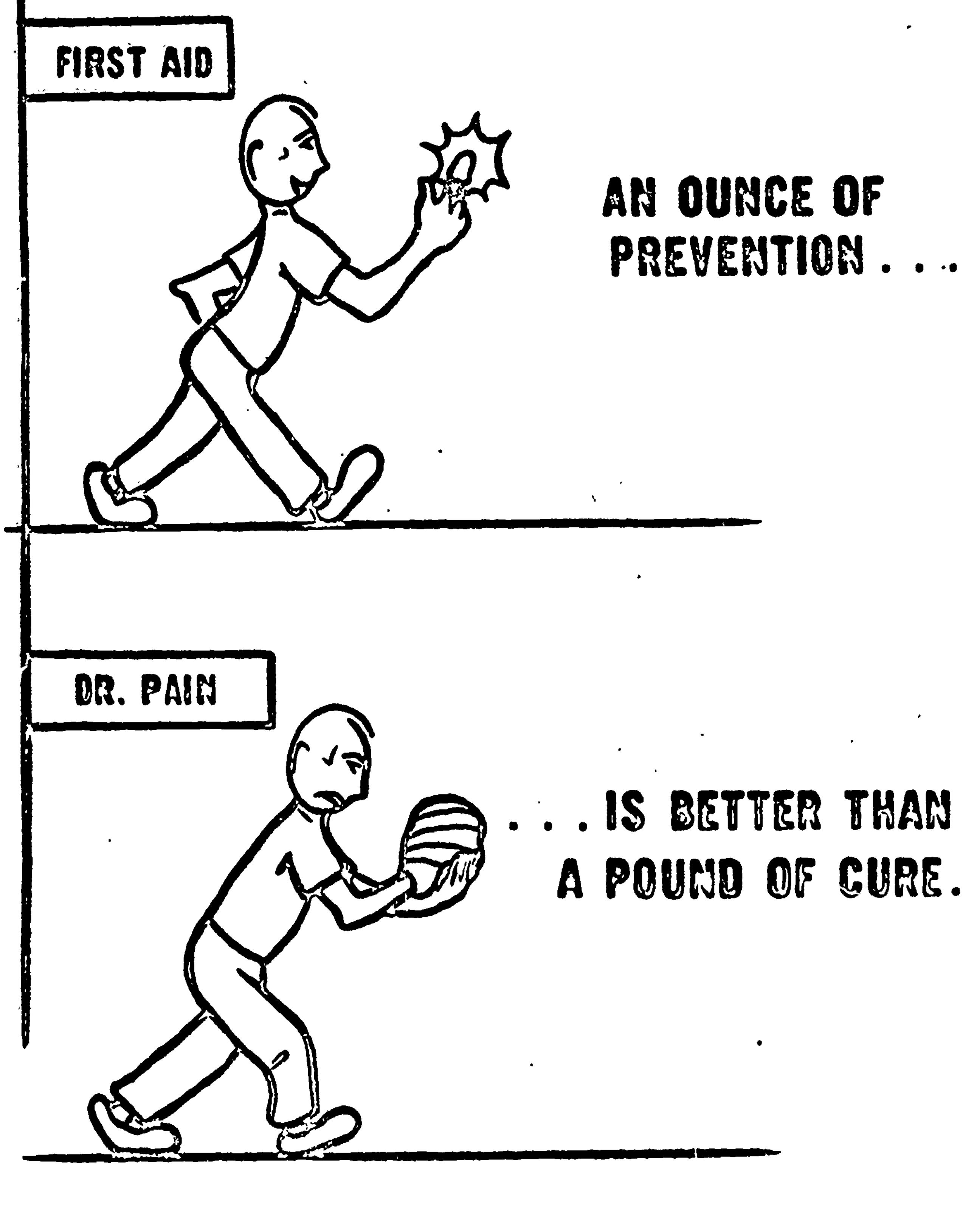
MAY CAUSE INJURY



A SHOP

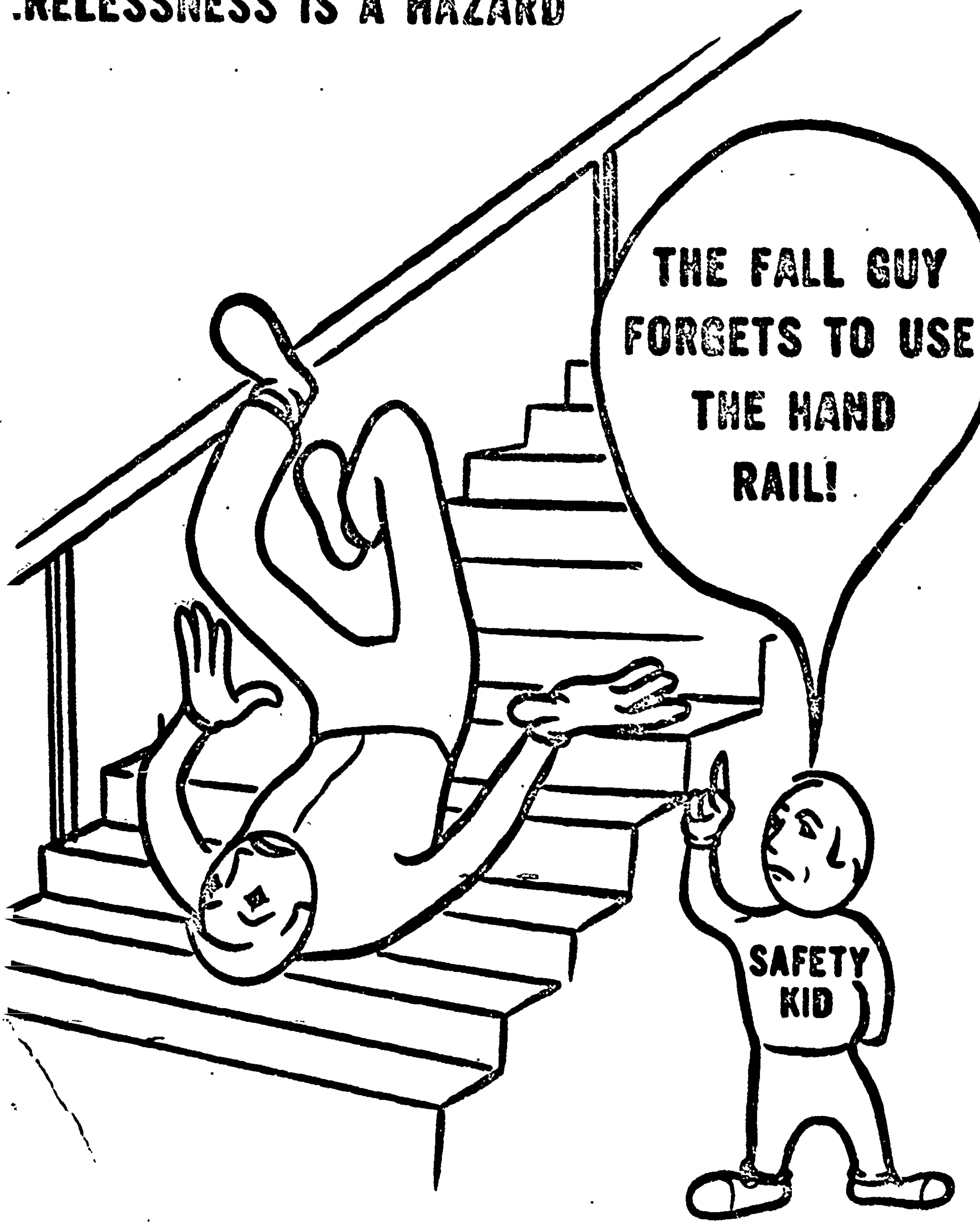
FIRST-AID KIT

PREPARATION

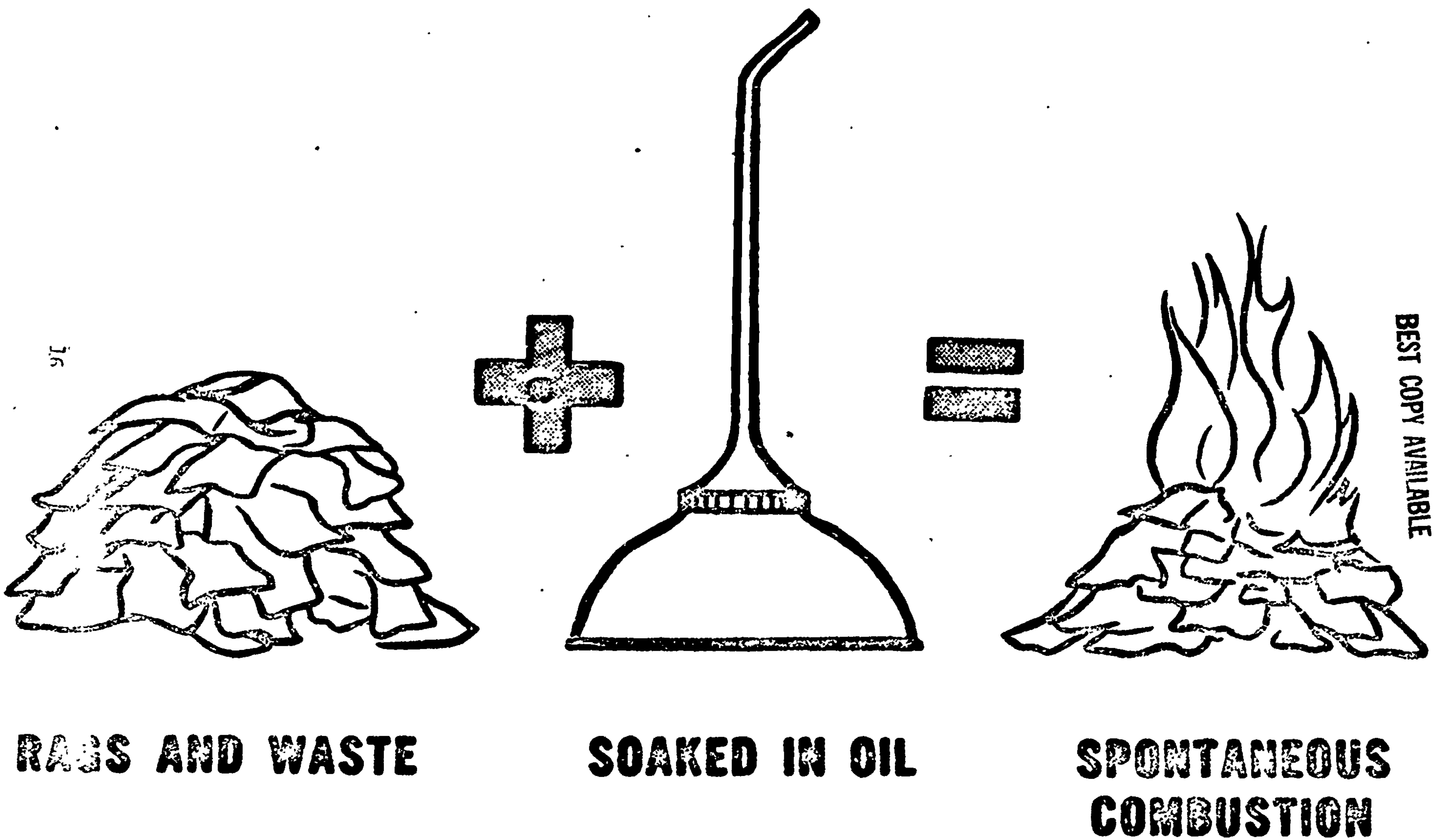


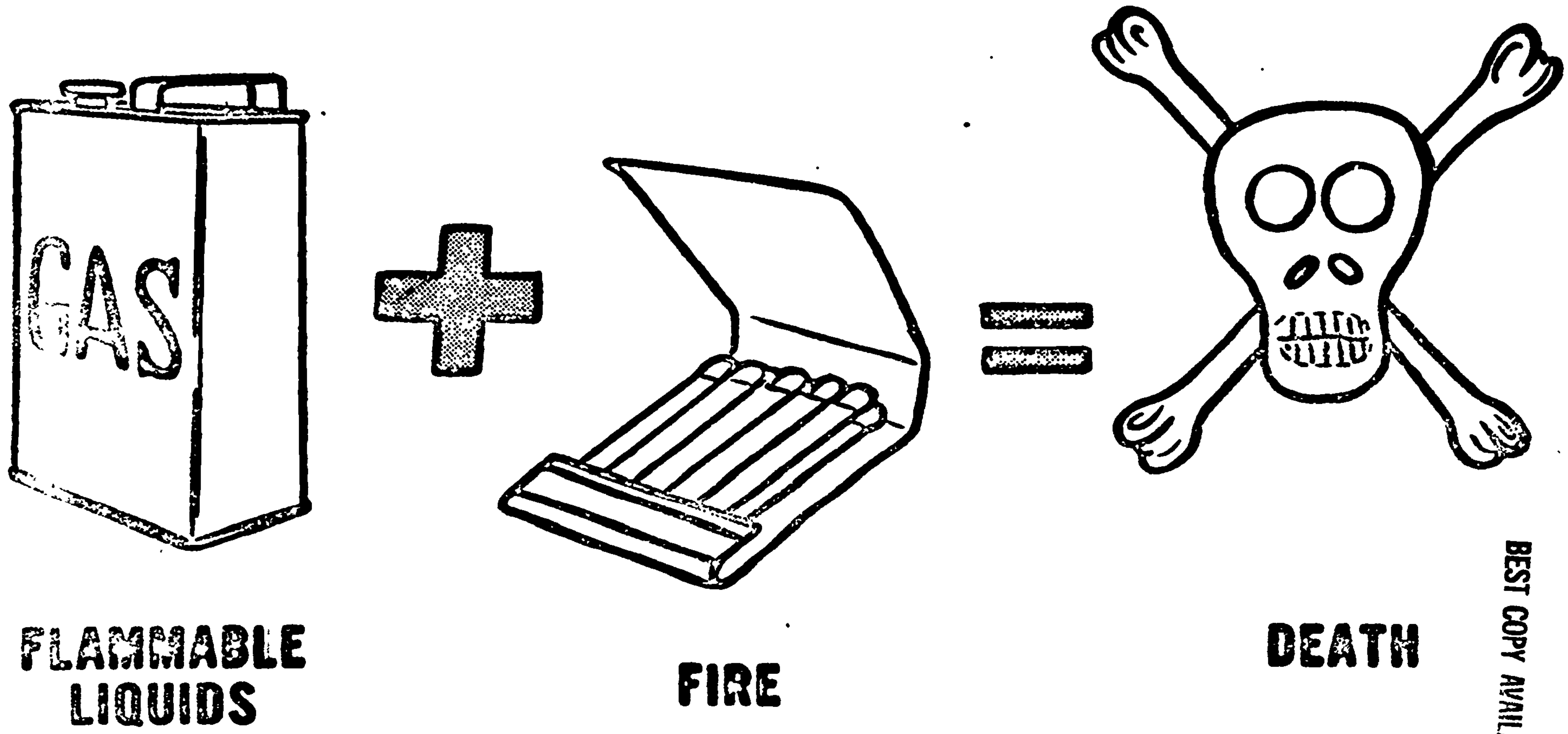
GET FIRST AID FAST

.RELESSNESS IS A HAZARD

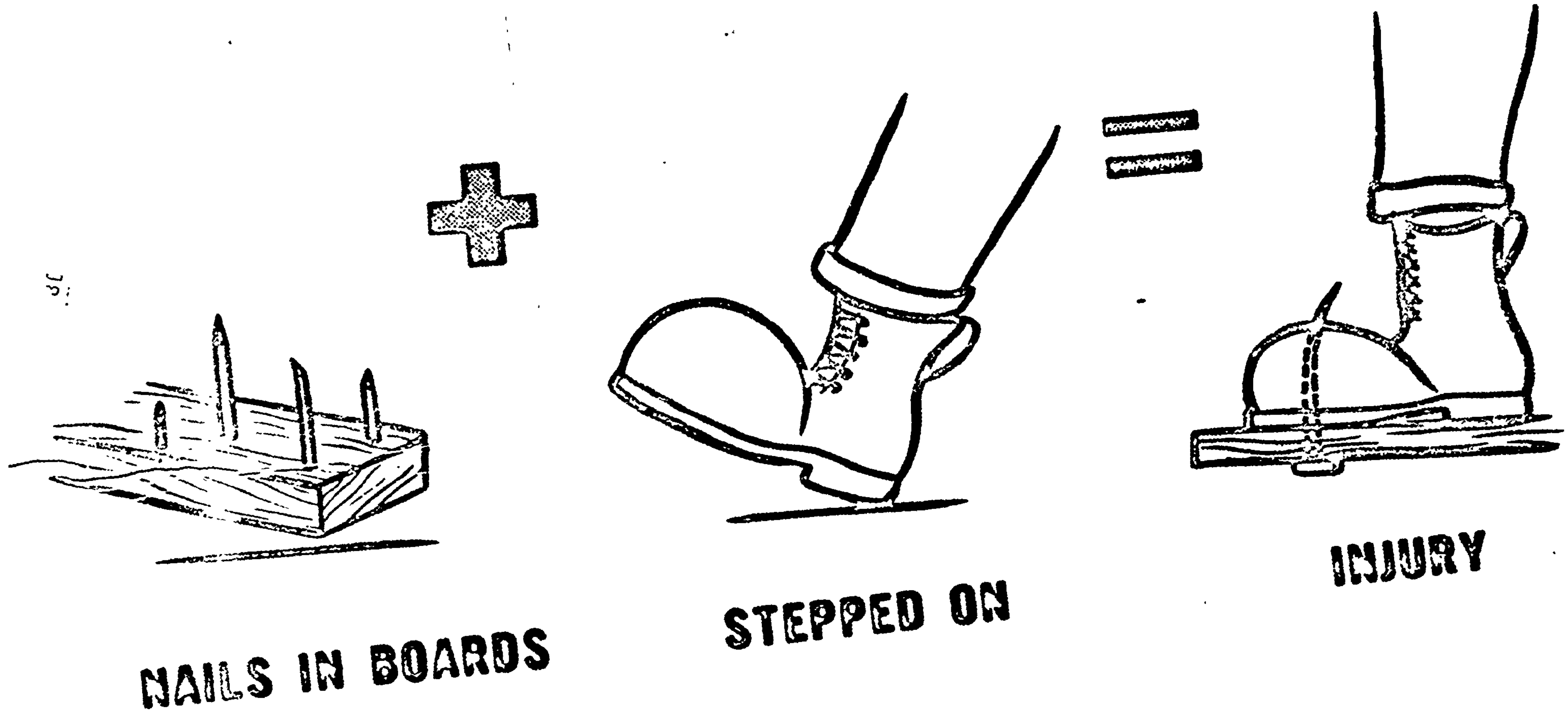


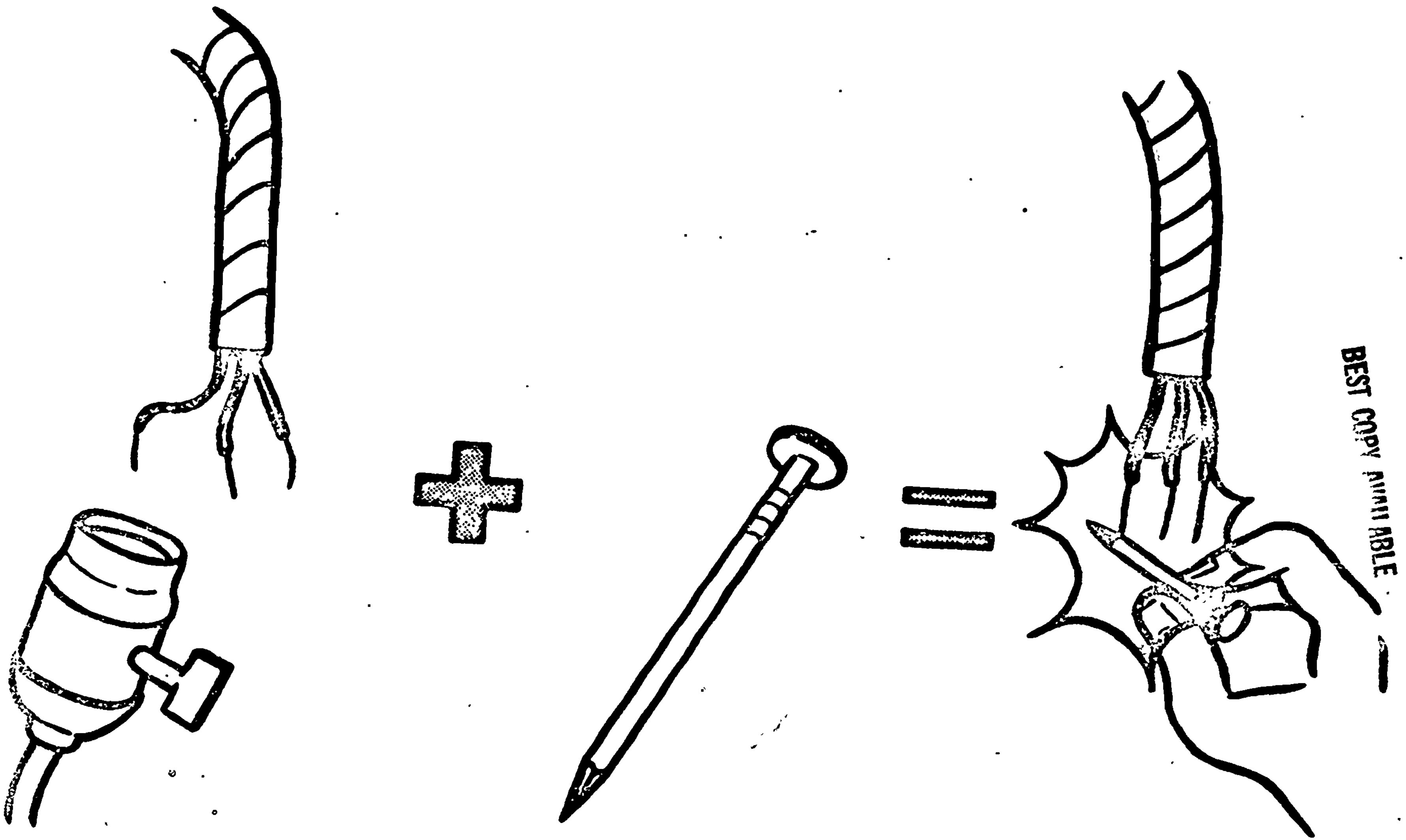
THINK AHEAD FOR SAFETY!


BEST COPY AVAILABLE
RAGS AND WASTE
SOAKED IN OIL
SPONTANEOUS
COMBUSTION

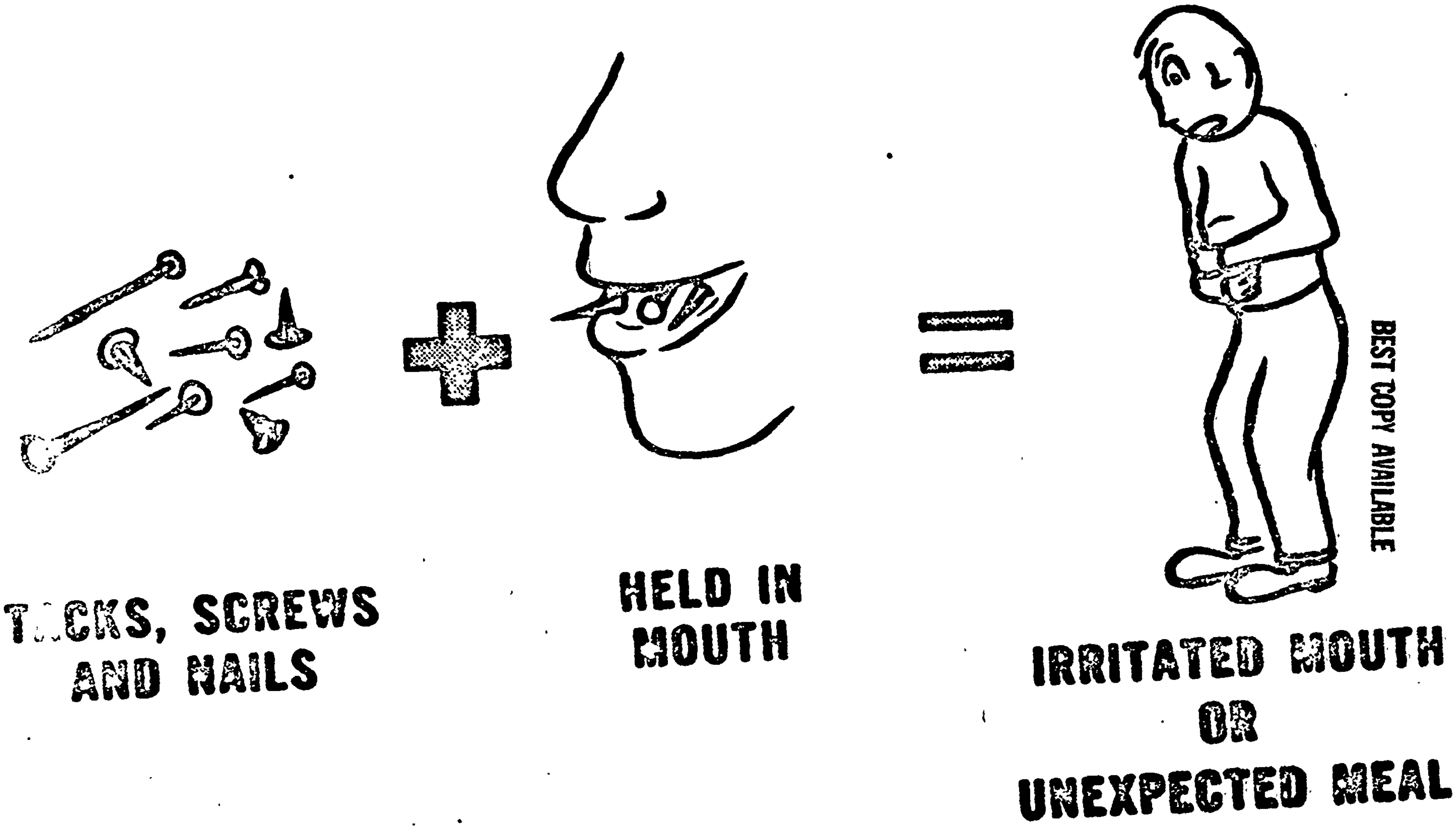


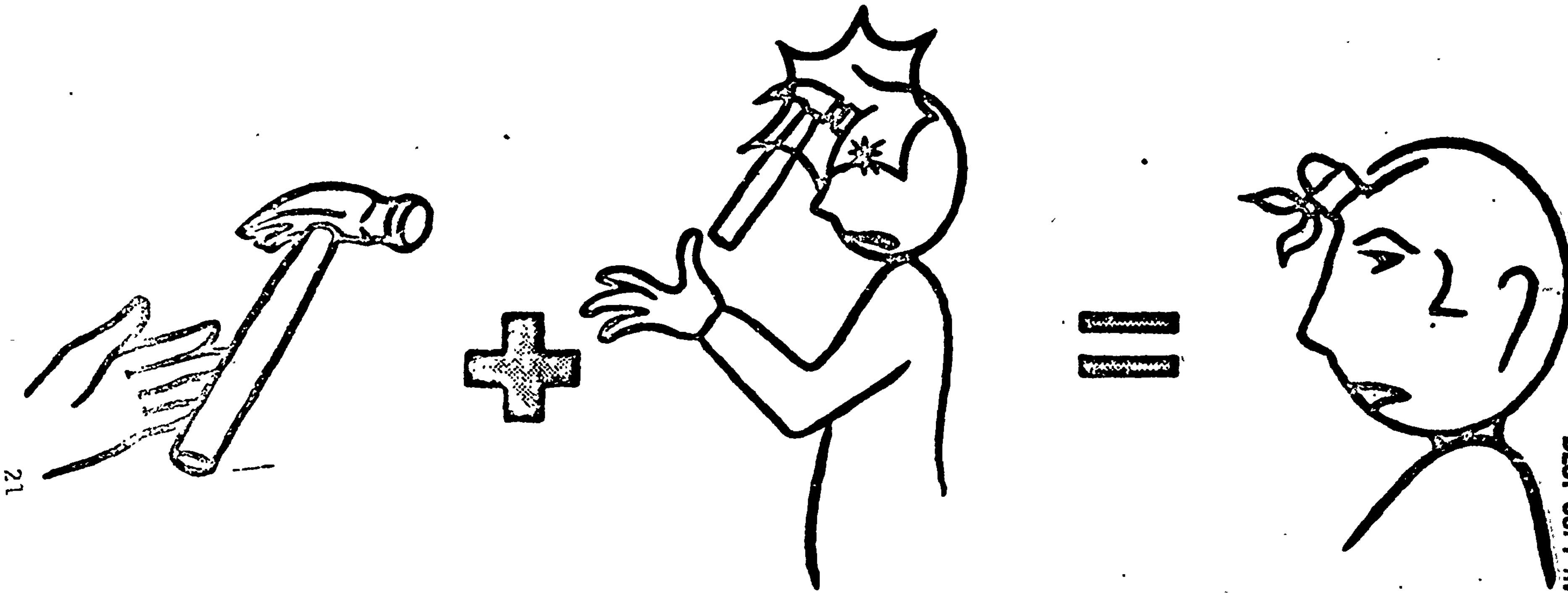
INFLAMMABLE MEANS FLAMMABLE!!


NAILS IN BOARDS
STEPPED ON
INJURY

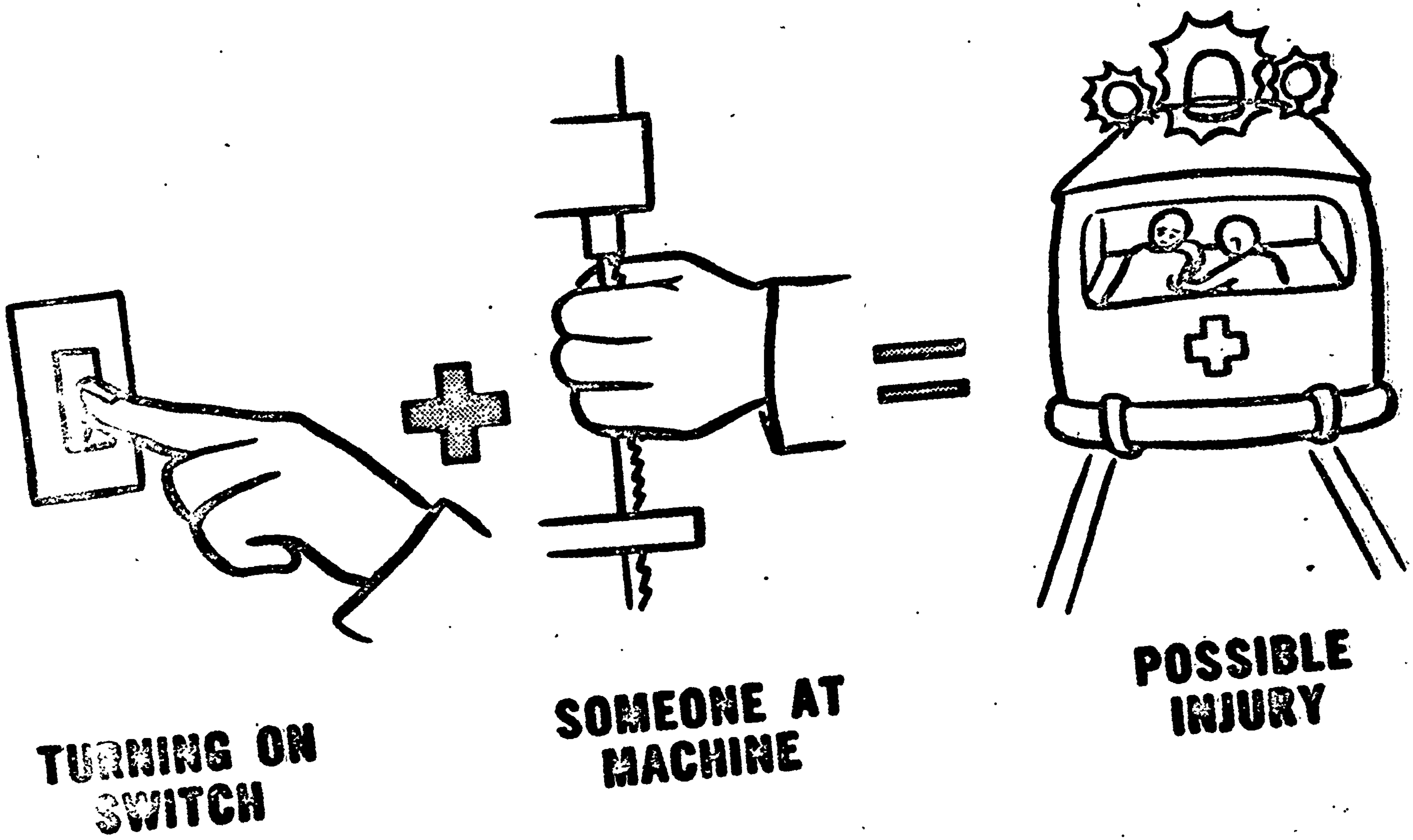


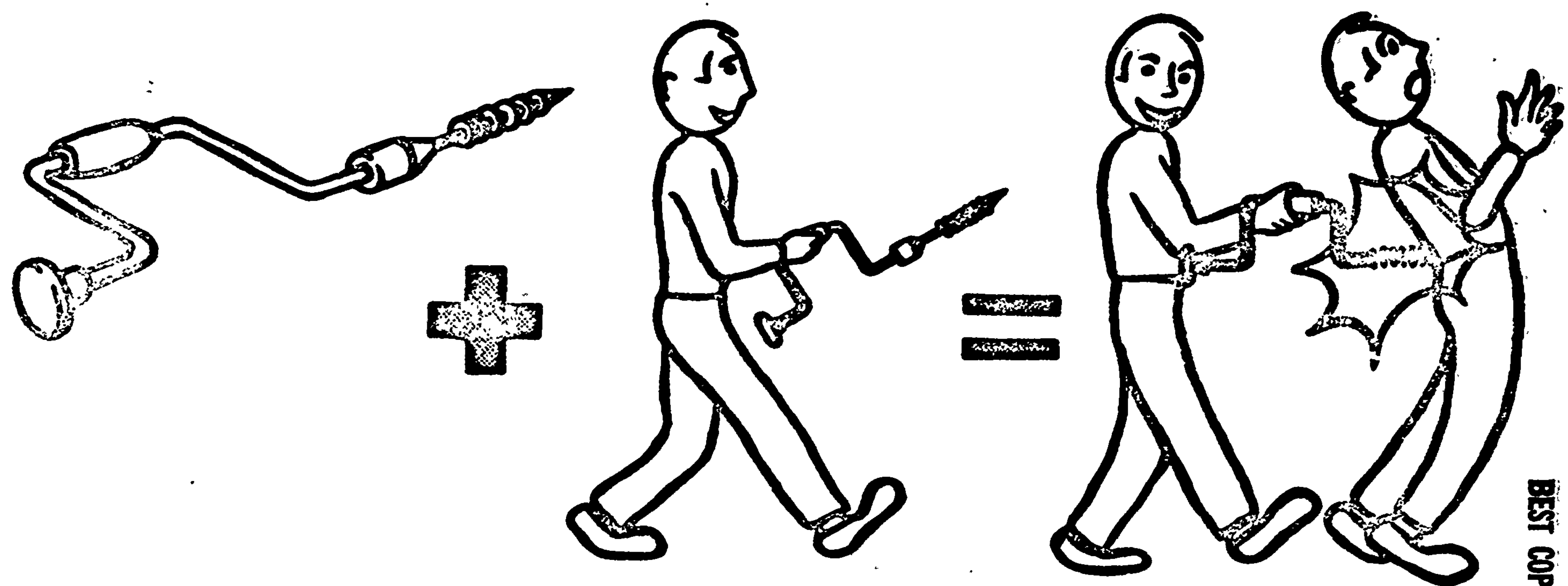
LIVE WIRES AND SOCKETS + TOUCHED WITH METAL OBJECT = ELECTRIC SHOCK


T..CKS, SCREWS
AND NAILS
HELD IN
MOUTH
IRRITATED MOUTH
OR
UNEXPECTED MEAL

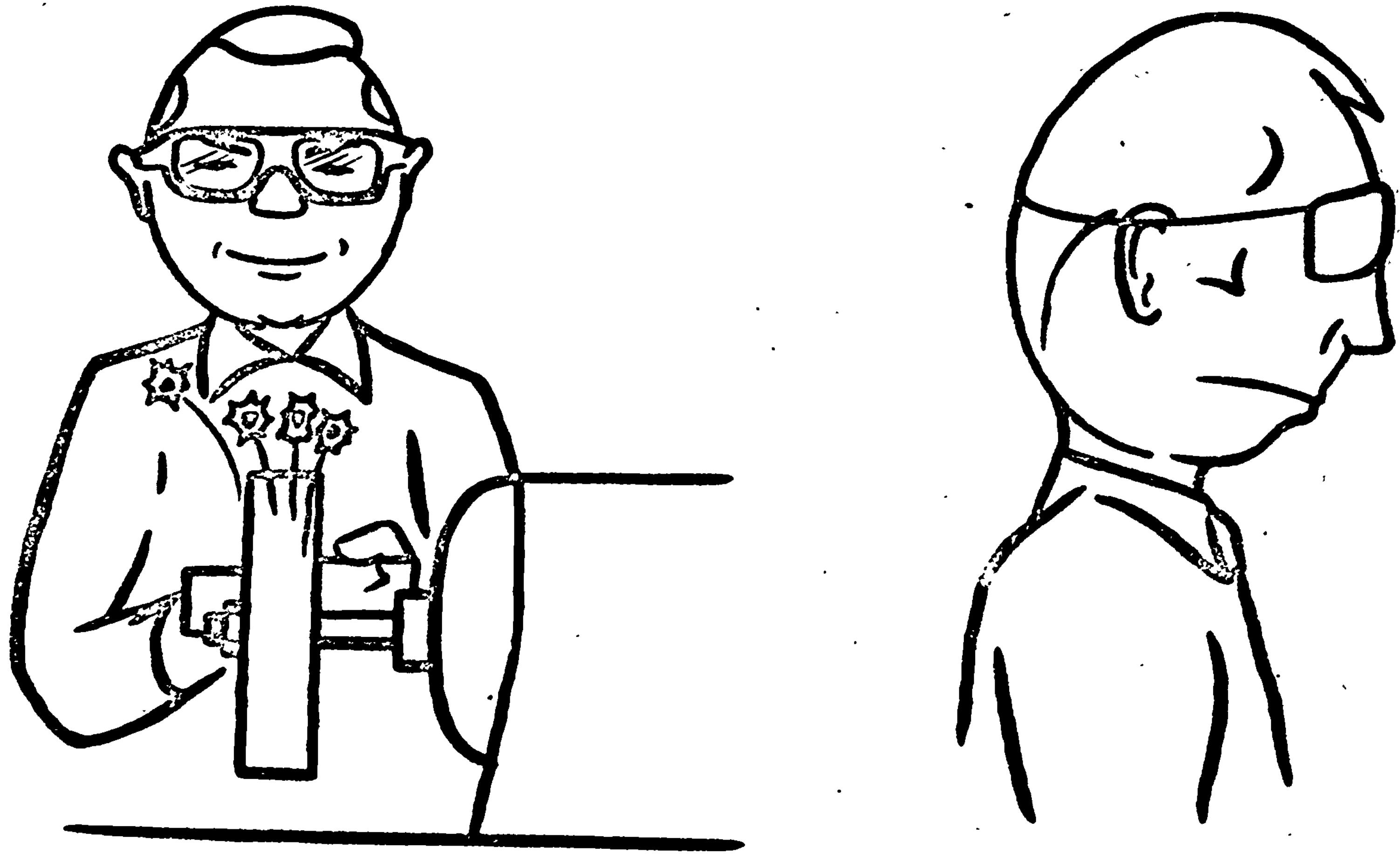


THROWING TOOLS TO OTHERS INJURY

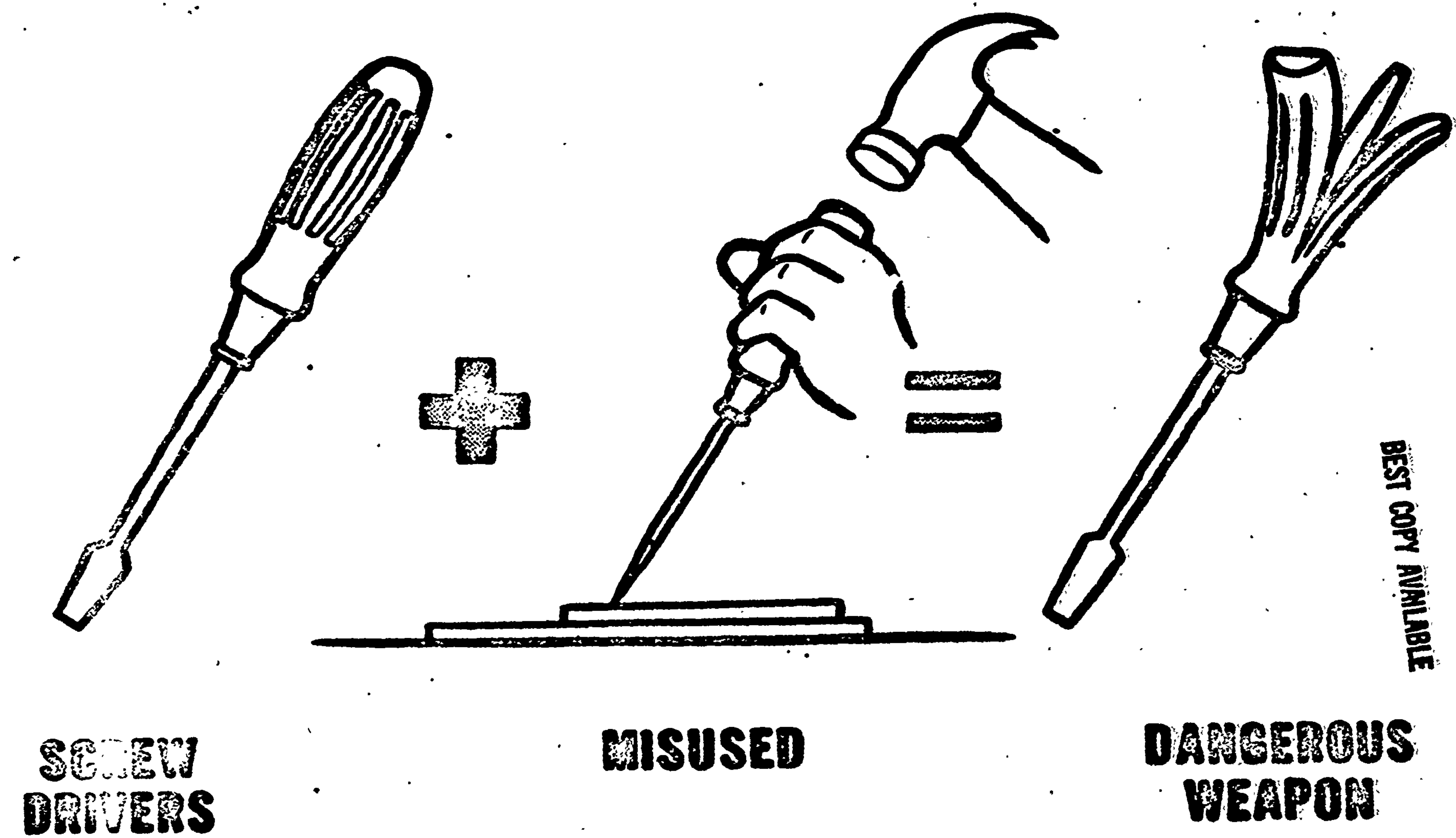

TURNING ON SWITCH
SOMEONE AT MACHINE
POSSIBLE INJURY

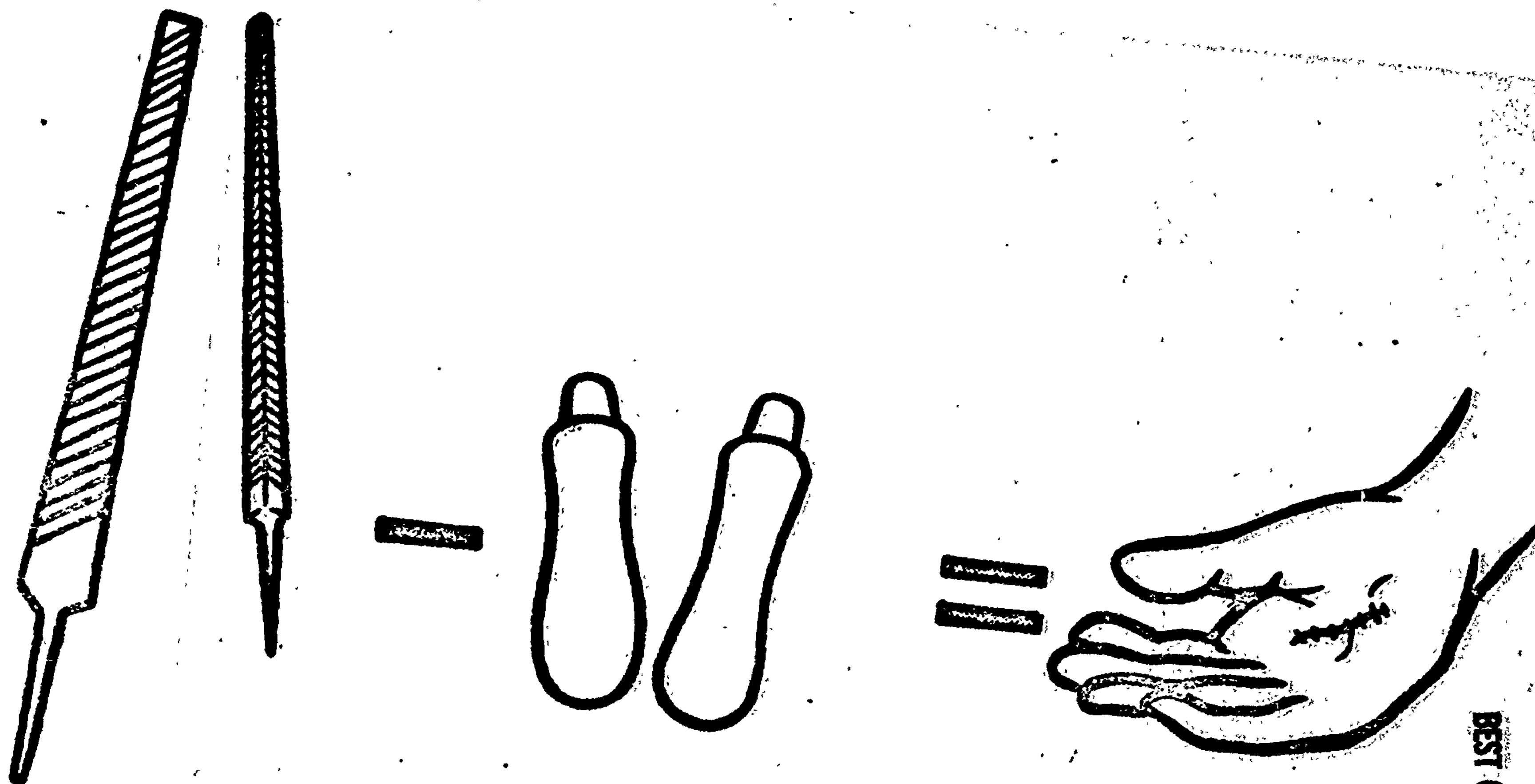


TOOLS + CARRIED IMPROPERLY = DANGER



EYES ARE HARD TO GET NOWADAYS!
PROTECT YOURS!!


SCREW DRIVERS
MISUSED
DANGEROUS WEAPON



FILES – **WITHOUT HANDLES** = **HAND INJURY**

BEST COPY AVAILABLE

TITLE: HAND TOOLS

UNIT: BENCHWORK

OCCUPATION: MACHINIST

OBJECTIVE: To Familiarize the student with the common hand tools and their correct use.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 2, pages 23-53.

DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. Describe and give the use of the most common types of hammers used by a machinst.
2. What are the principal parts of a hammer head?
3. What is a drift punch and how is it used?
4. What type of punch is used to drive out dowel pins, rivets and cotter pins?
5. How does a prick punch differ from a center punch?
6. What is the point angle of a center punch?
7. What tool is used to mark lines on metal?
8. List the common types of screw drivers.
9. How should the blade of a worn screw driver be ground?
10. Describe the following spanner wrenches.
    (a, Pinhook. (b) Pin-faced. (c) Hook.
11. What type of wrench is used on socket head set screws?
12. What is the point angle of a flat cold chisel?
13. List and describe the four common types of cold chisels.
14. Explain how to heat - treat a chisel.
15. Why must goggles be worn when chipping?

TITLE: HACKSAWS AND SAWING

UNIT: BENCHWORK

OCCUPATION: MACHINIST

OBJECTIVE: To acquaint the student with proper use of the hacksaw and the types of hacksaw blades.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 2, pages 35-39.

DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. How is the length of a hacksaw blade determined?
2. Why does a blade have set?
3. What is the principal factor when selecting the number of teeth on the blade for a job?
4. What are the advantages of a flexible back blade?
5. What is the minimum number of teeth that should be in contact with the work when sawing?
6. What pitch blade would be best for sawing 1" diameter bar steel?
7. What pitch blade would be best for sawing thin sheet metal?
8. What is the correct speed for sawing?
9. What are the common causes of broken blades?
10. What are slotting blades?

TITLE: FILES AND FILING

UNIT: BENCHWORK

OCCUPATION: MACHINIST

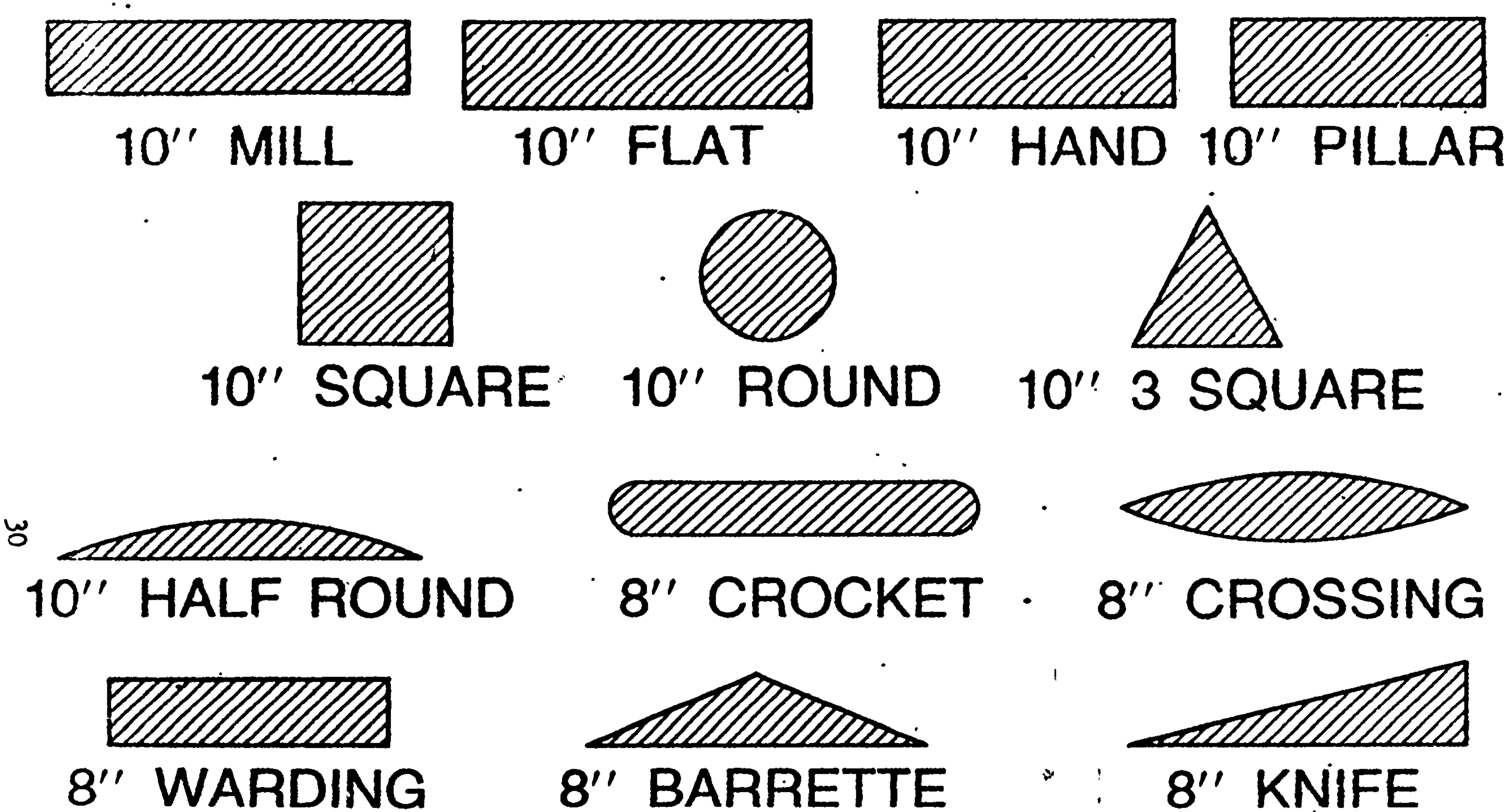
OBJECTIVE: To acquaint the student with the types and proper uses of files.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 2, pages 39-46.

DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. Name and sketch the different shapes of files.
2. What are the two classes of files?
3. What are the six grades of files?
4. Why should a file never be used without a handle?
5. What is draw filing?
6. When filing why cross the stroke?
7. What is the preferred file for draw filing?
8. What file is best for aluminum or other soft metals?
9. What are needle files?
10. How are files cleaned?



KINDS OF FILES

TITLE: MEASURING AND LAYOUT TOOLS

UNIT: BENCHWORK

OCCUPATION: MACHINIST

OBJECTIVES: To acquaint the student with the types and uses of measuring and layout tools.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 3, pages 54-99.

DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. Name the parts of a combination set.
2. What are the standard graduations on a machinist rule?
3. What is the advantage of a hook rule?
4. Explain the use of dividers.
5. What are inside calipers?
6. What are outside calipers?
7. Explain the use of hermaphrodite calipers.
8. What is a surface plate?
9. What is a surface gage and how is it used?
10. List the common types of micrometers and give the use of each.
11. What are the five principal parts of a micrometer?
12. What are the graduations on a micrometer?
13. What is the smallest graduation on a vernier micrometer?
14. What are the graduations on a vernier caliper?
15. What are the advantages of a vernier caliper?
16. What is a vernier bevel protractor?
17. What is the smallest graduation on a vernier bevel protractor?
18. What are gages used for?

19. What are gage blocks and how are they used?

20. What is a sine bar?

21. What is a dial indicator?

22. What is a snap gage?

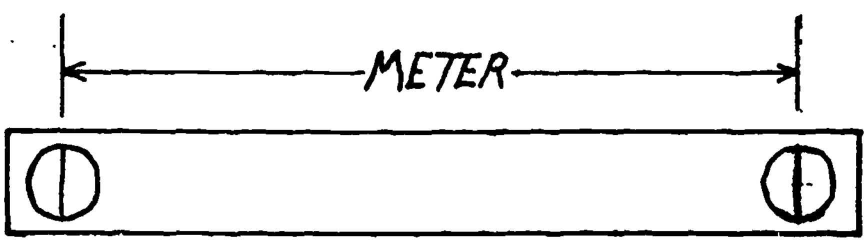
23. What are the three classes and accuracy of gage blocks?

24. What is a master gage?

25. What is a radius gage?

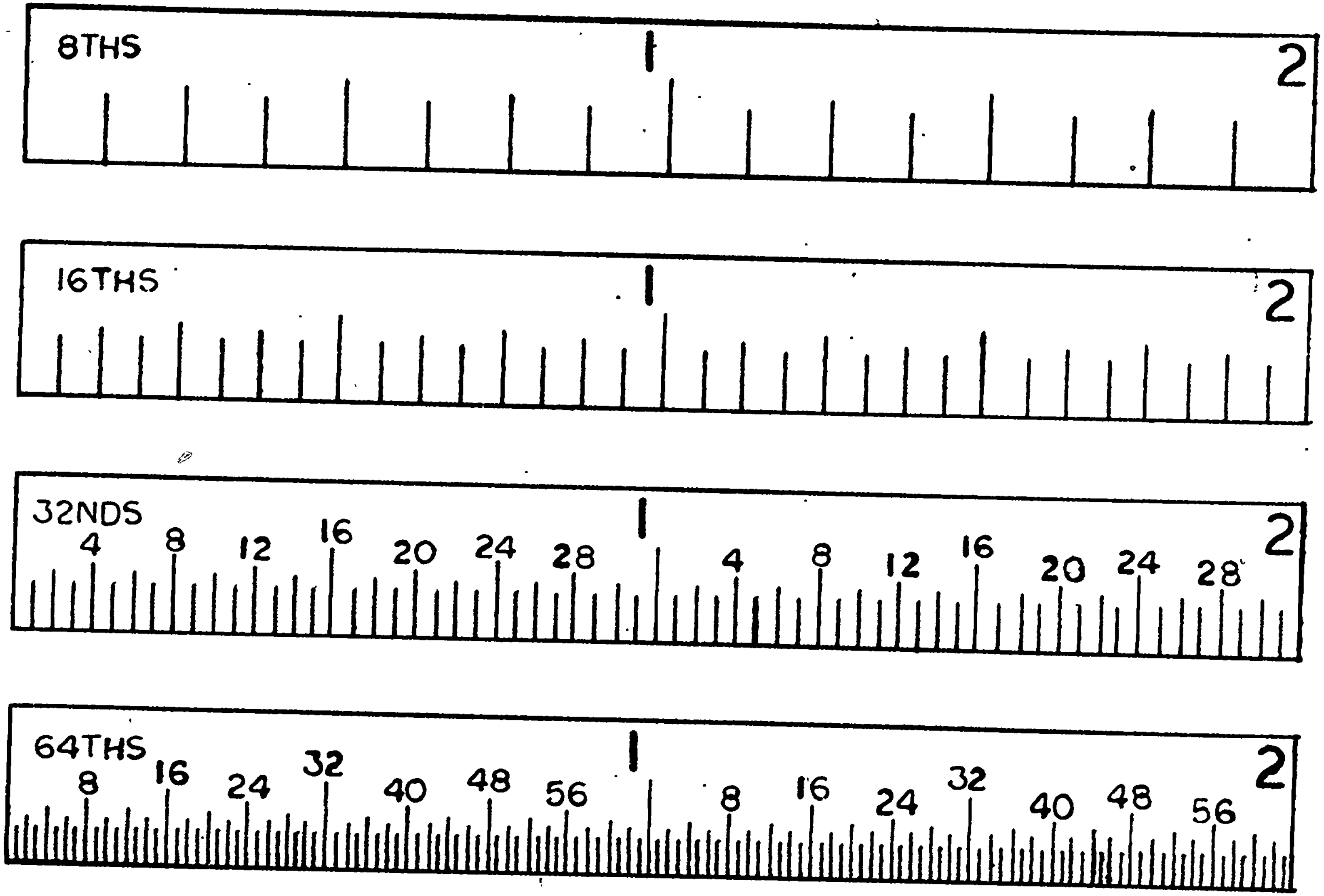
$\frac{3{,}600}{3{,}937}$ OF A METER OR 0.9144

AN INCH = $\frac{1}{36}$ OF A YARD

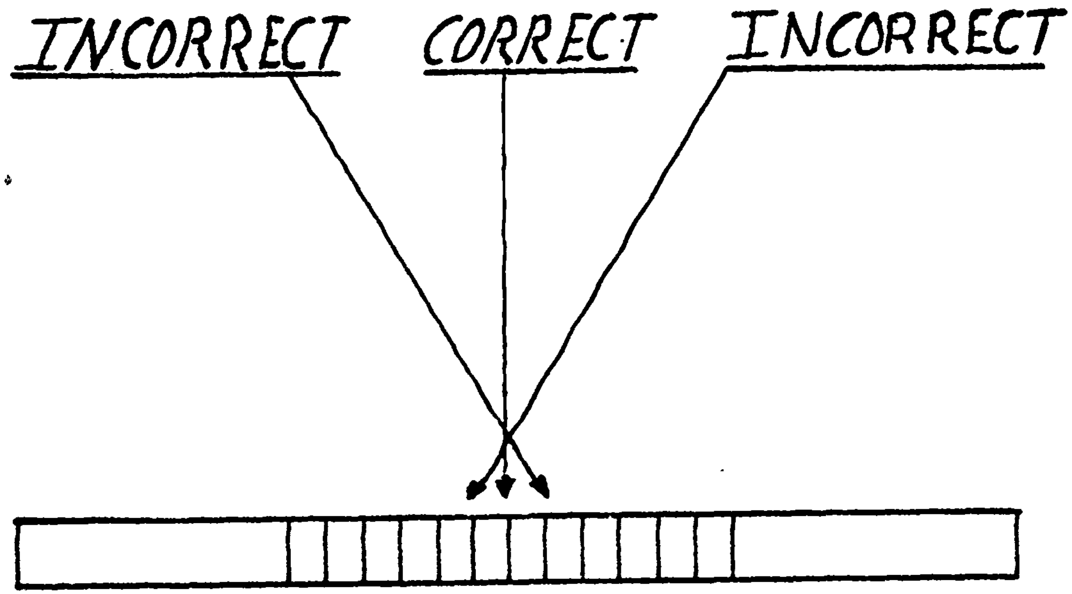


METER = 39.37 INCHES

RULE



8THS
1
2
16THS
1
2
32NDS
4 8 12 16 20 24 28
1
4 8 12 16 20 24 28
2
64THS
8 16 24 32 40 48 56
1
8 16 24 32 40 48 56
2



PARAllAX OR OBERVATIONAL ERROR

USE OF THE STEEL RULE

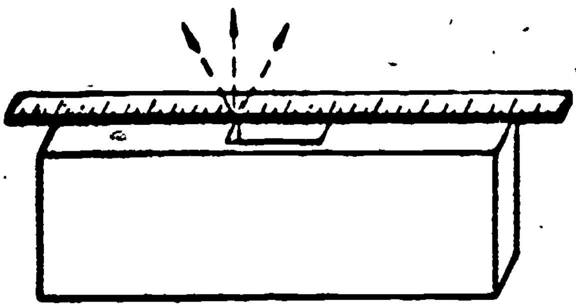
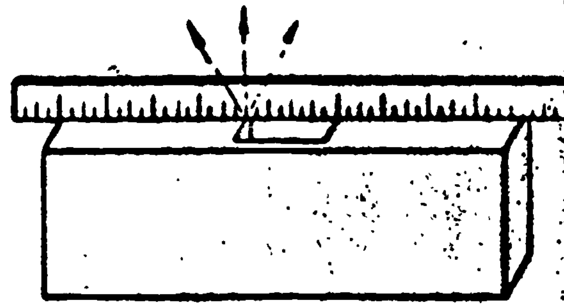


FIGURE I



Parallax or observational error is the apparent movement of an object as viewed from varied positions. On the left in Figure I, parallax may very well affect the measurement. Parallax will not affect the measurement on the right because the graduation of the scale lies directly on the reference point, therefore, even though the object is viewed from varied positions, the same reading will be obtained. Parallax may also be overcome by aligning one's head so that the line of sight is directly over the measured point.

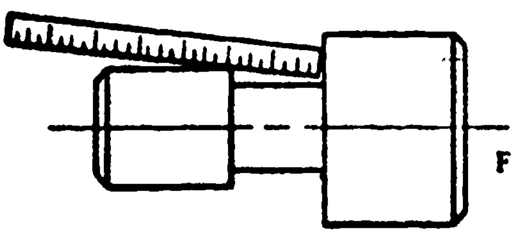
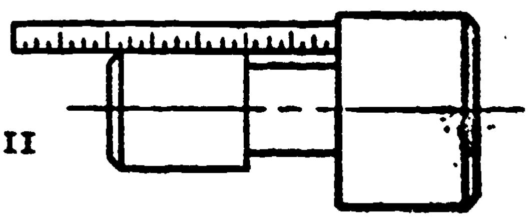


FIGURE II



When measuring, one must be certain they are working from the proper reference surfaces and along the correct line of measurement. It is obvious on the left in Figure II that the scale of the rule does not lie along the line of measurement, and incorrect readings will be obtained. On the right in Figure II, the shoulder acts as a suitable reference surface to measure the work piece.

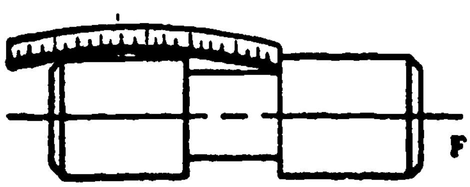
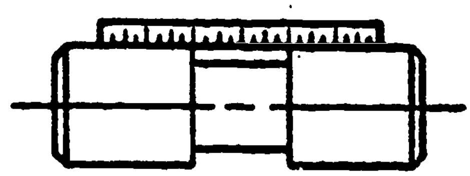


FIGURE III



One must never bend or distort a rule in striving to obtain a more advantageous reference point as on the left in Figure III. With the rule bent in this manner, it is impossible to obtain a correct measurement. The correct procedure in measuring this work piece is shown at the right. An inch graduation is used as a reference point with the scale edge lying directly on the line of measurement. In this manner parallax may be avoided and a reliable measurement is obtained.

BEST COPY AVAILABLE

USE OF THE STEEL RULE

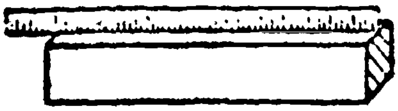
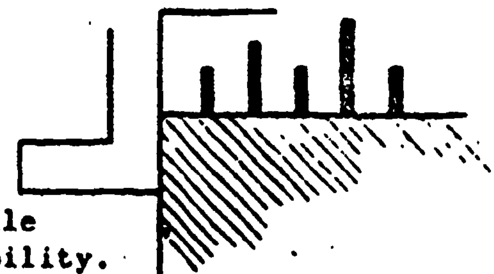


FIGURE I

Difficulty may be encountered in attempting to align the end of the rule with the edge of the work. End wear on the work or rule would make alignment more difficult, and precision and reliability would be lost.

FIGURE II



The hook rule provides a reference point within its construction. This type of rule provides for greater precision and reliability. One must make certain that wear between the hook and rule, or burrs on the work piece are not influencing precision and reliability.

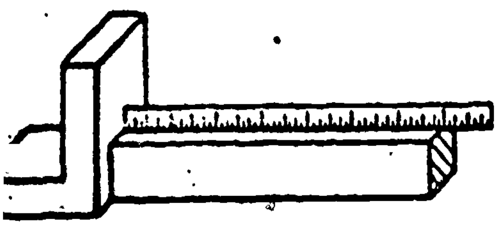
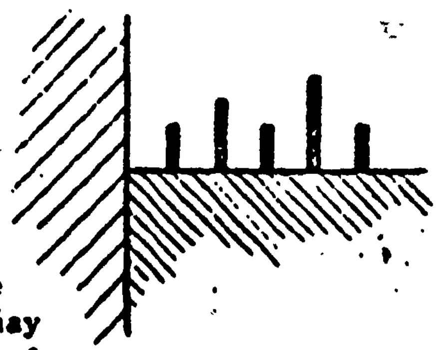


FIGURE III



With the work piece placed firmly against a reference surface such as the angle plate in this figure, precision and reliability may be achieved. The angle plate acts as the reference point for the work and eliminates manipulation errors. It is essential that the work have a good bearing against the plate, that no burrs are present, and that wear on the end of the rule is not affecting reliability.

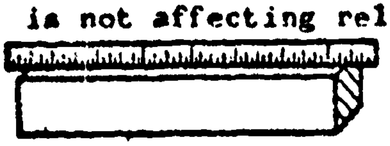
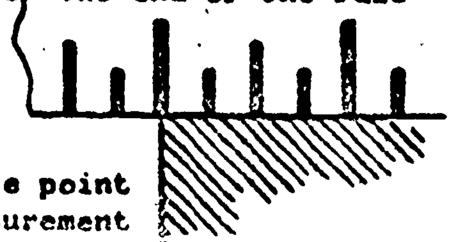


FIGURE IV.



By using an inch graduation as a reference point on the scale, precision and reliable measurement are possible with the rule. With the graduation directly on the edge of the work, parallax may be avoided and by not using the end of the rule, end wear is inconsequential.

USE OF THE DEPTH GAGE AND COMBINATION SQUARE

FIGURE I

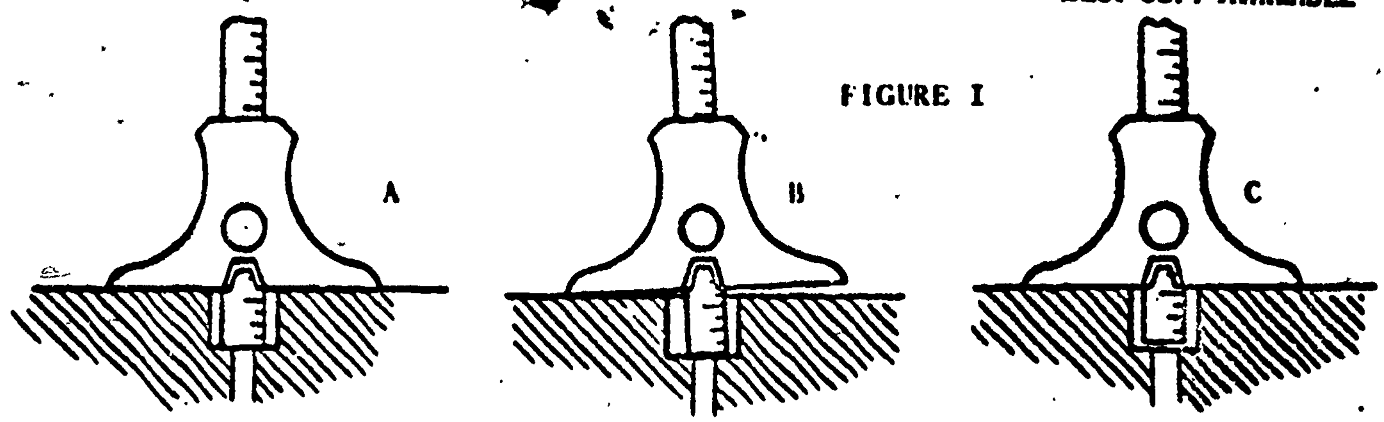
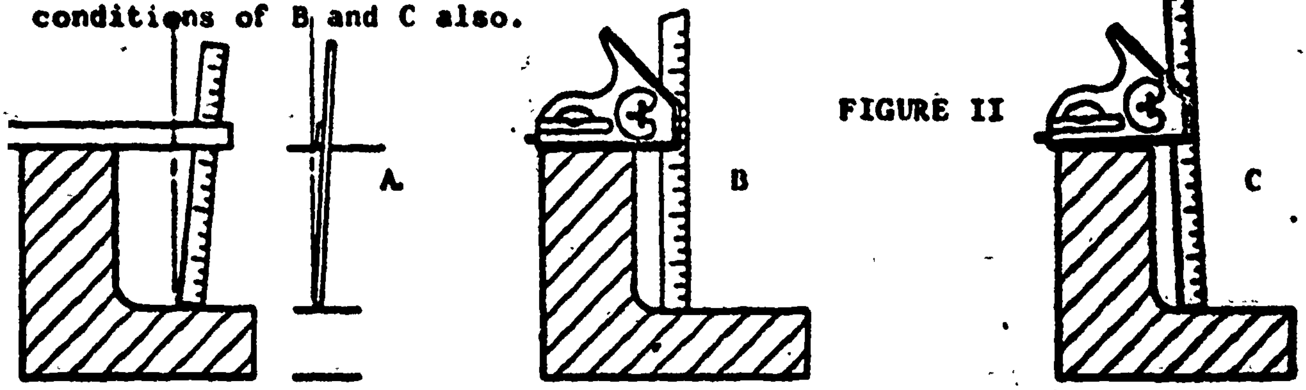


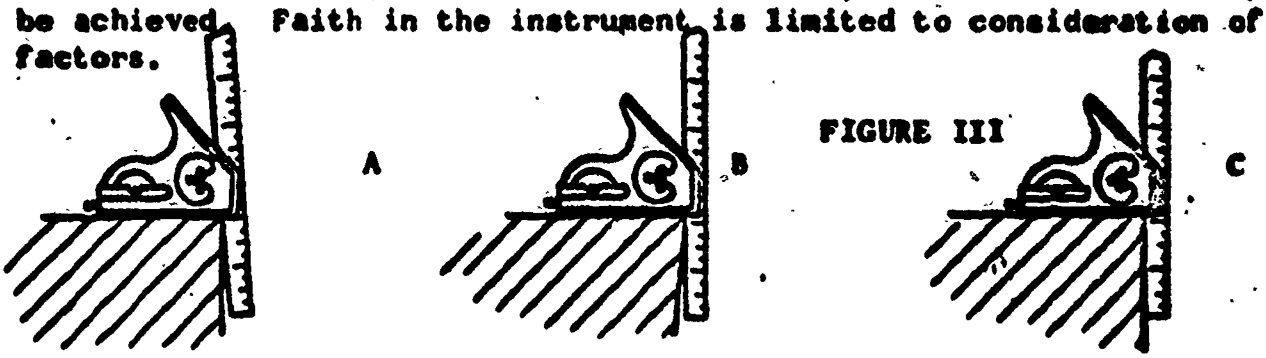
Figure I-A illustrates the proper use of the depth gage. The faces of the head have good bearing on the reference surface, and the reference surface of the head is perpendicular to the axis of the hole. Figure I-B and I-C illustrate manipulative errors which may occur, and it is apparent in these figures that wear on either the face of the head or end of the rule would affect reliability. Carelessness or foreign matter could very well produce conditions of B and C also.

FIGURE II



A, in Figure II, illustrates the difficulty involved in trying to obtain a reading in a haphazard manner; alignment is difficult with just a rule and piece of straight stock. Figure II-C shows how manipulative errors due to presence of dirt, burrs, or improper use may occur. The head in B has a good bearing on the reference surface, therefore reliability may be achieved. Faith in the instrument is limited to consideration of wear factors.

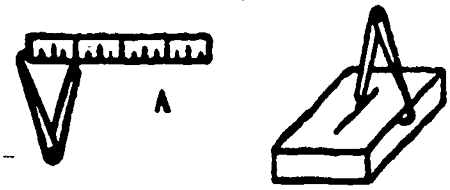
FIGURE III



Wear between the head and the rule can produce the conditions in A of Figure III, whereby the work is square but the instrument is not. Figure III-B illustrates proper use of the combination square as does C. In B however, the work is out of square. Conditions somewhat similar to those found in A and B could occur, if there were poor reference surface conditions present.

USE OF THE HERMAPHRODITE CALIPERS

FIGURE I



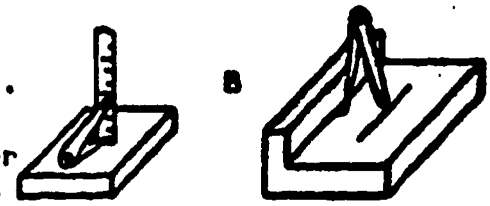
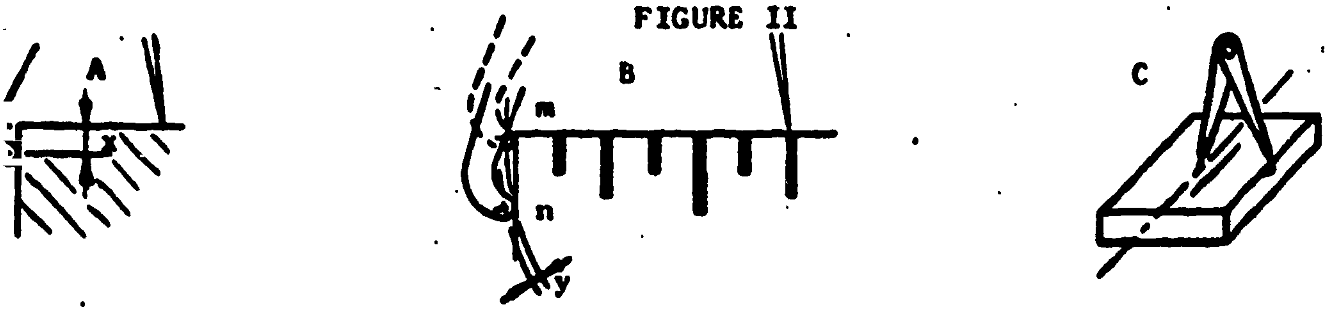
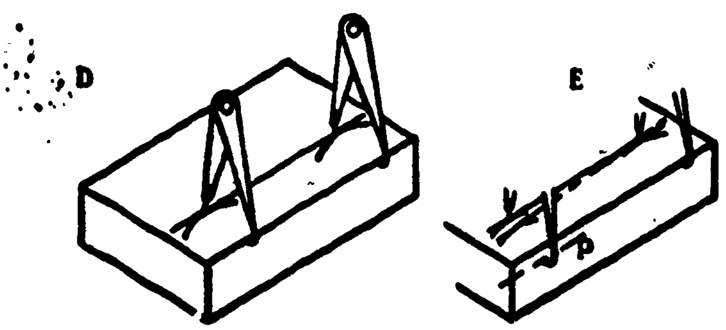


Figure I shows very common errors in application of the hermaphrodite calipers. Manipulation errors would be made quite easily under these conditions. The caliper leg would have to move in a straight line along the edge of the work and at an equal distance down from the edge of the work as on the scale in A. In both examples false measurement would be encountered, if the line between the caliper leg and divider leg were not perpendicular to the line one desired to scribe.

FIGURE II





In Figure II the distance x in A must be constant in measuring if reliability is to be maintained. The distance y in B illustrates the variation that occurs with the caliper leg at points m and n. Figure II-C illustrates the variation encountered when the conditions of B are present or when the line between the caliper and divider leg is not perpendicular to the line to be scribed. Figure II-D illustrates a more reliable method of scribing a line parallel to the work edge. The dotted line in E points out the error which would occur if the caliper leg should slip to position p.

FIGURE III

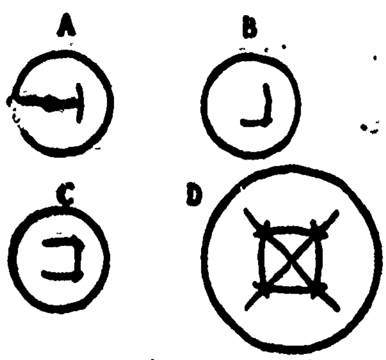
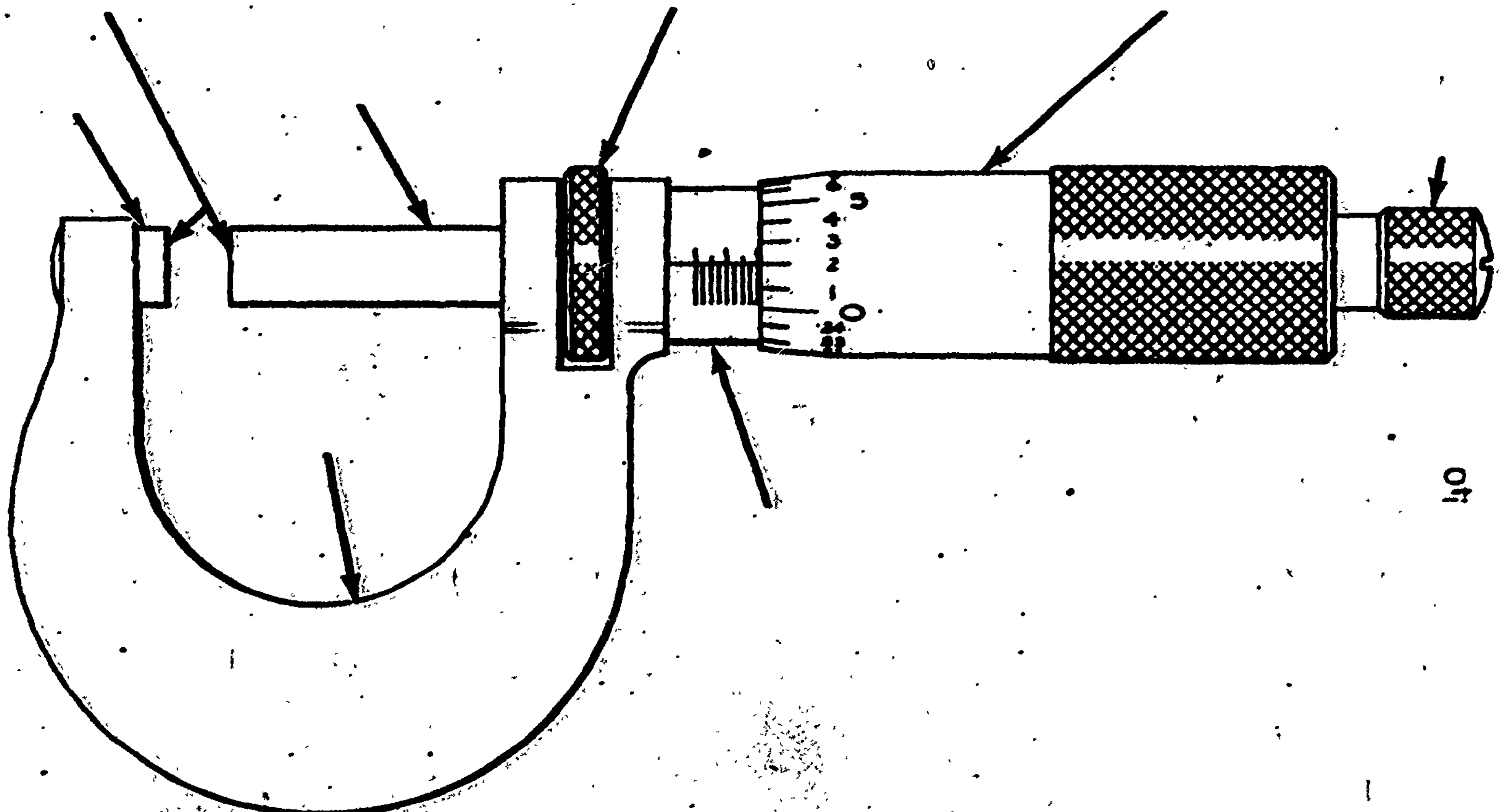


Figure III shows the sequence of layout of the center of cylindrical work with the hermaphrodite calipers. A series of arcs are scribed from varied points on the diameter of the work. The center of these arcs is found to be the center of the work. This is a common application of the hermaphrodite calipers, and if care is observed, reliable measurement may be attained.



PARTS OF THE MICROMETER

MEASURING SURFACES

LOCK NUT

THIMBLE

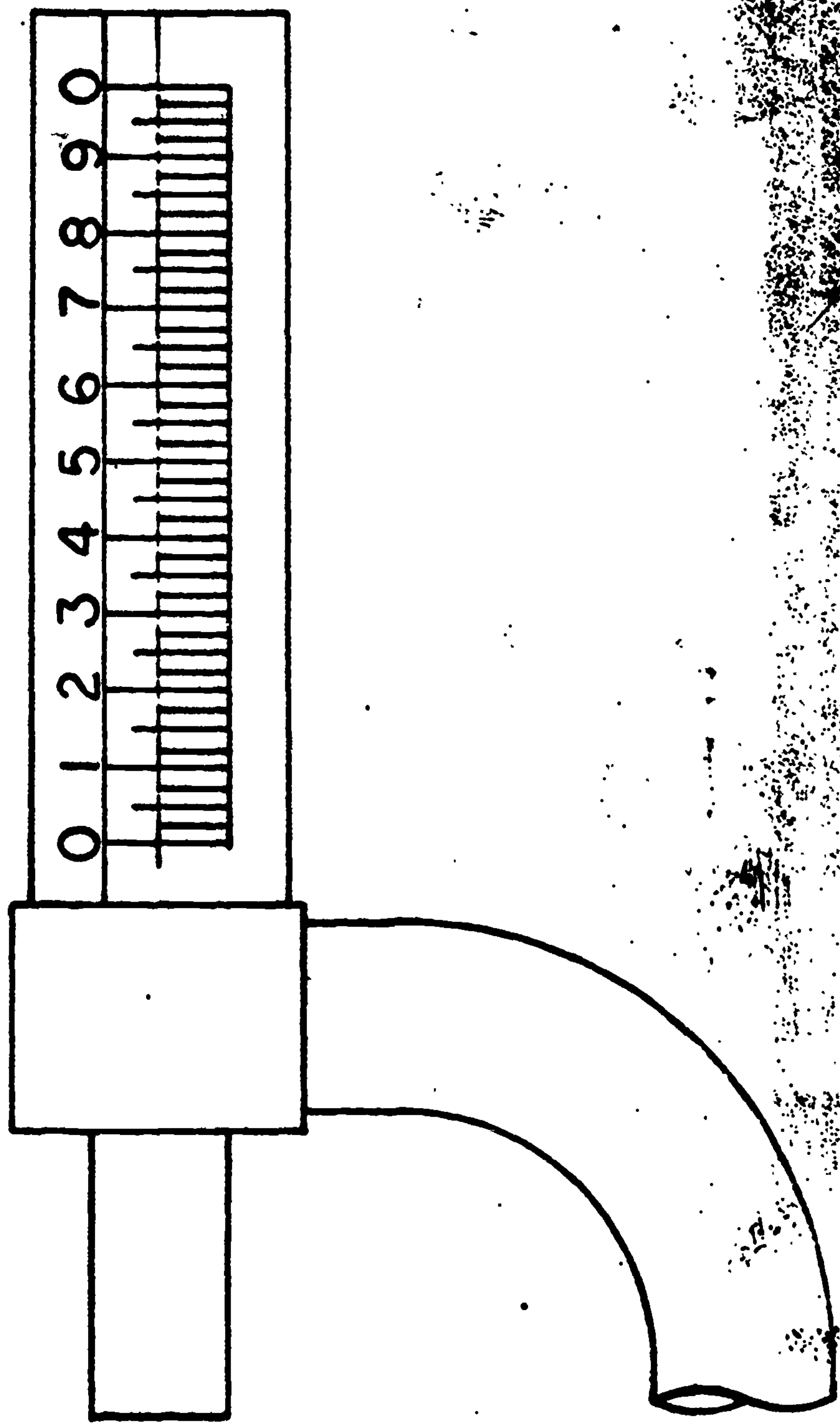
ANVIL

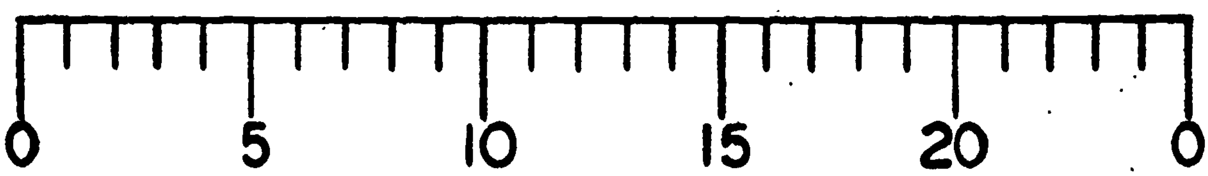
SPINDLE

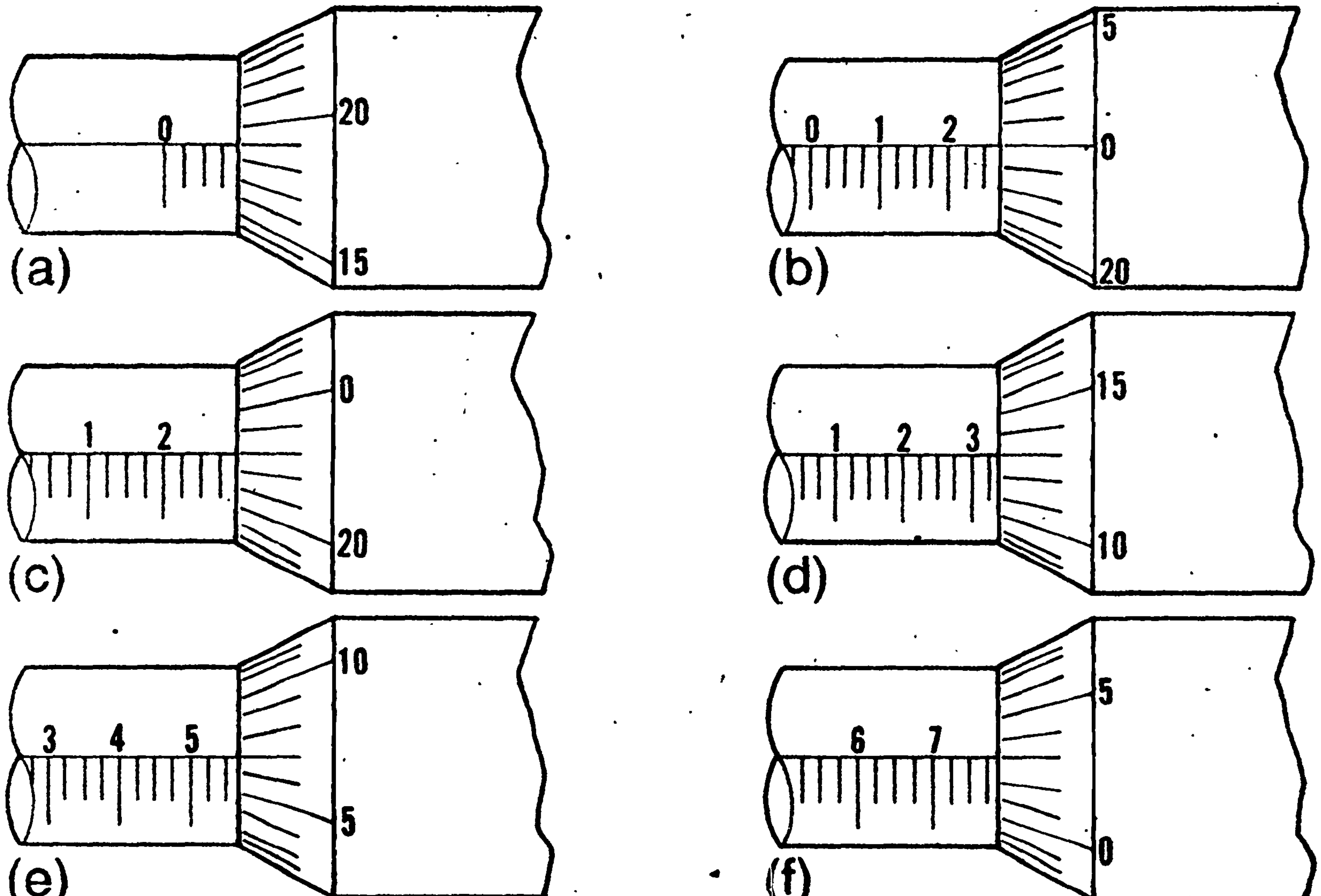
RATCHET STOP

FRAME

SLEEVE

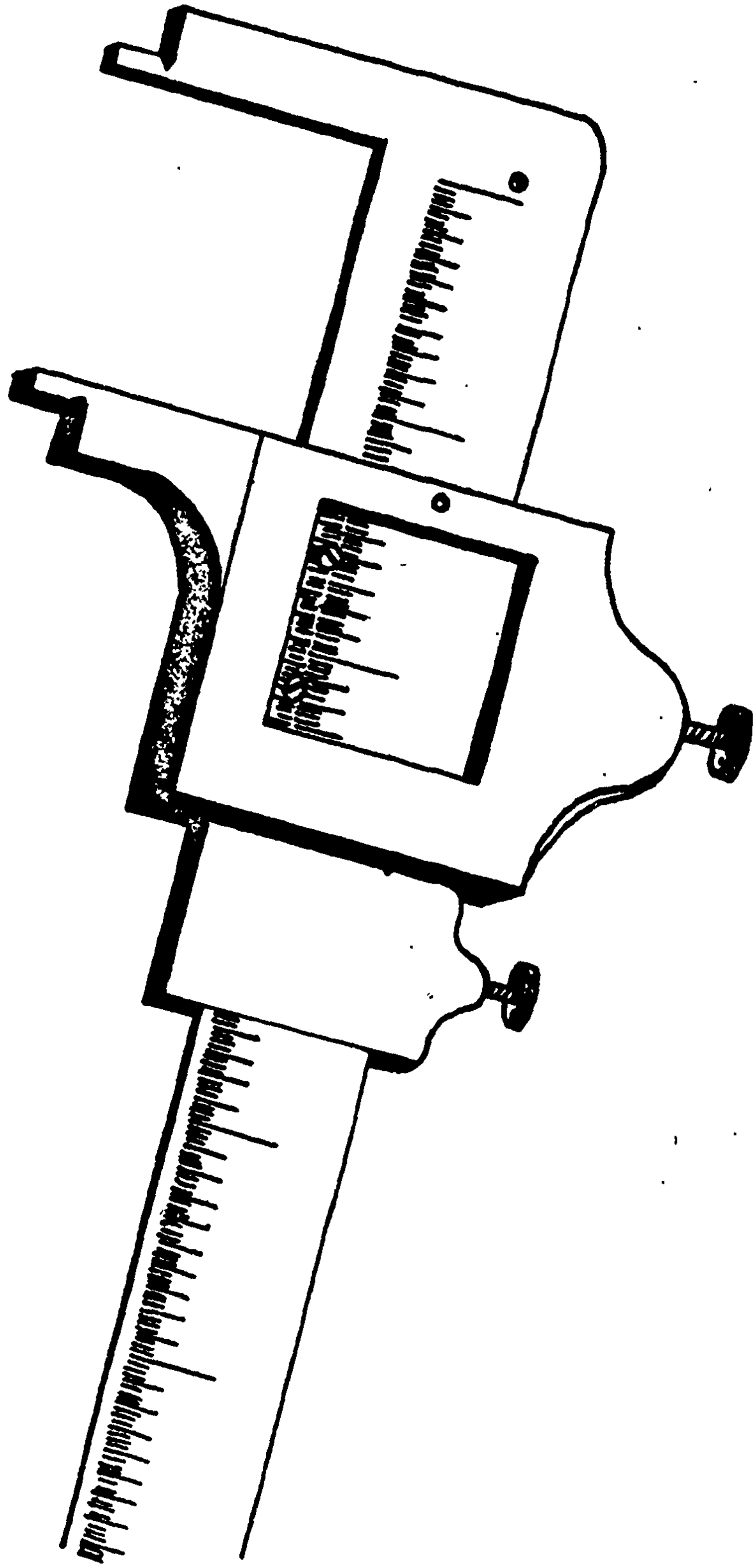

0 1 2 3 4 5 6 7 8 9 0


0
5
10
15
20
0



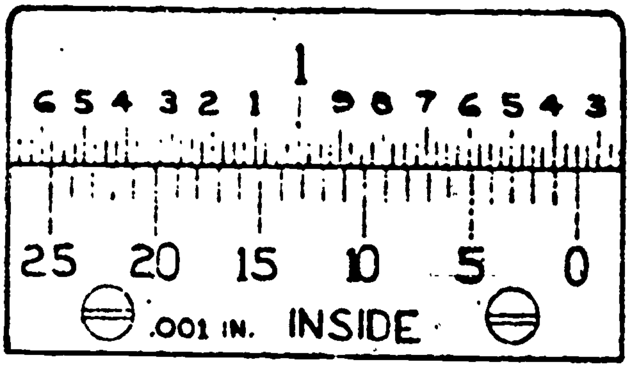
MICROMETER READINGS

VERNIER
CALIPER



VERNIER CALIPER FEATURES

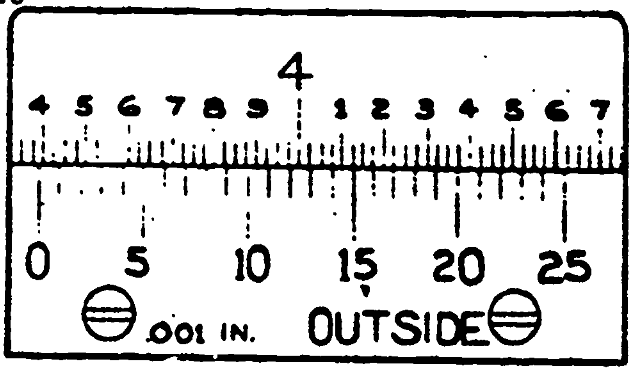
The scales of the vernier caliper are illustrated at the right, with the top view showing the inside scale and the second view showing the outside scale. These serve to illustrate the basic features of any vernier scale essentially. The bottom portions of the illustrations represent the vernier scale which moves relative to the basic scale to indicate divisions of that scale. The inside scale serves several purposes here. It not only shows that with inside measurements the scale must be read from right to left, but it also illustrates the instance where more than two lines will be in coincidence. This occurs with exact .025" division readings, in this case .350". The outside on the other hand, must be read from left to right, and the reading in this instance is 3.392".

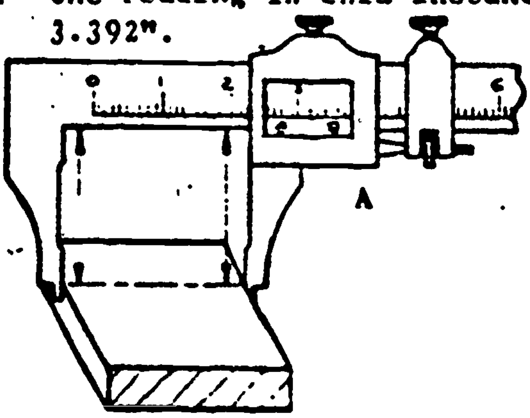


A

FIGURE I

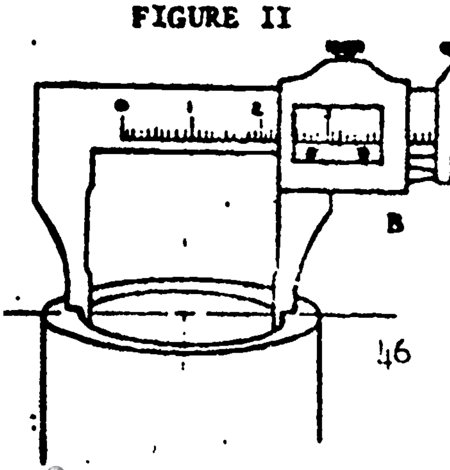
B





A

FIGURE II



B

Manipulation errors may occur through misuse of the vernier caliper. This happens due to the fact that the design of the instrument is such that it depends a great deal upon the skill of the user. Alignment and "feel" are two very important considerations if reliability is to be attained. Proper alignment of the vernier caliper along the line of measurement as illustrated in Figure II, is essential if precision measurements are to be obtained. In this illustration the jaws of the vernier caliper must be exactly perpendicular to the line of measurement, and the imaginary line between the two measuring surfaces must be one and the same with the line of measurement. These factors are considered and maintained in Figure II, thereby assuring reliability and precision to a large degree. "Feel" is essential to determine the minimum separation of the jaws with outside measurement as in II-A and maximum separation of the jaws with internal measurements as in II - B.

USE OF THE VERNIER CALIPER

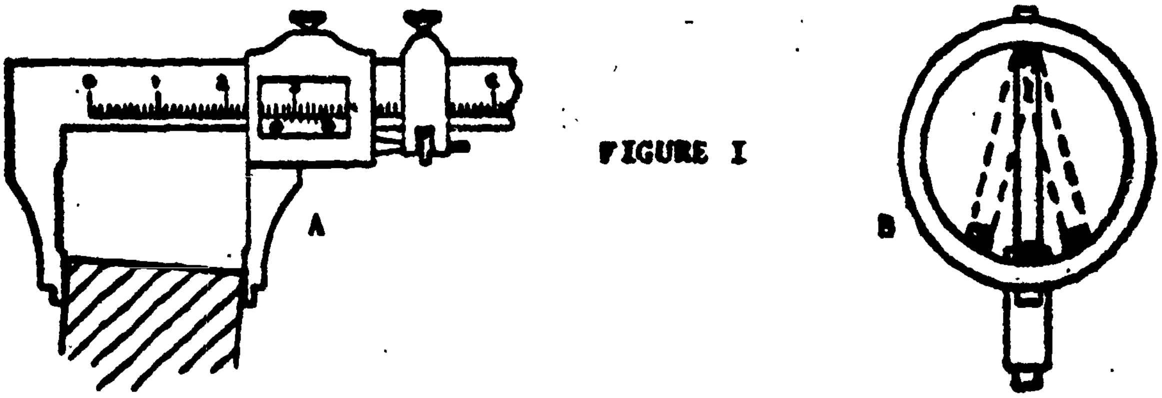


FIGURE I

Alignment and "feel" are manipulative factors which can affect reliable measurement. In Figure I alignment errors are illustrated whereby the line between the measuring surfaces is not parallel with the line of measurement or the jaws not perpendicular to the line of measurement. This is brought out in I-A, and one may readily see that the true dimension of the work piece is not being measured. The bottom view of a cylinder being measured in B of Figure I, serves to illustrate how it may be necessary to rotate the movable jaw across center several times, and determine by "feel" when the jaws lie along the line of measurement or the actual diameter of the work piece.

FIGURE II

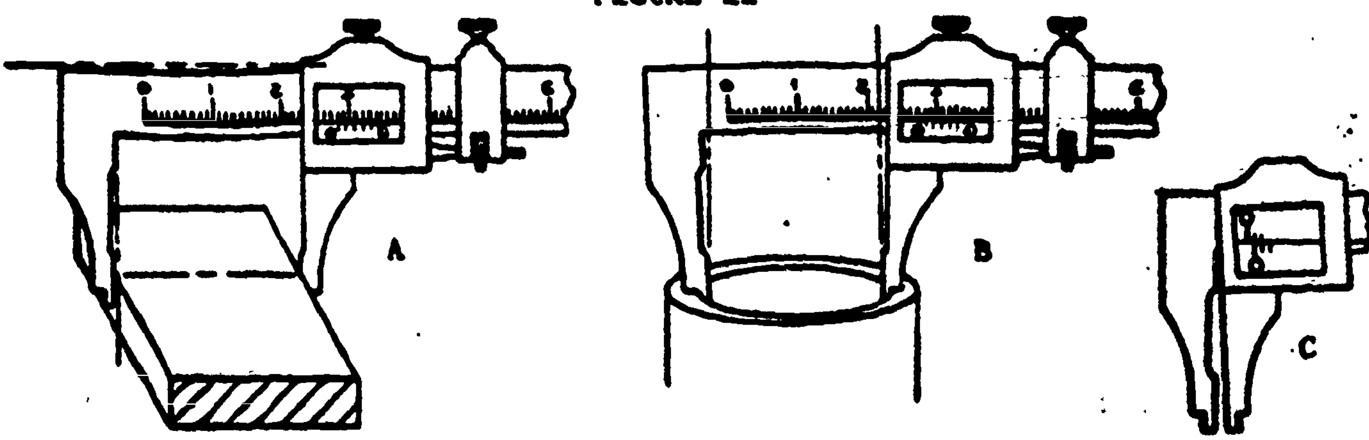


Figure II serves to illustrate the effects of improper usage of the vernier caliper. Excessive gaging pressure could cause the exaggerated conditions of II-A and II-B. With A, if too much force were applied, the movable jaw could bend slightly causing the vernier scale to move too far along the basic scale and a reading somewhat less than the actual dimension of the work piece would be obtained. With B, a somewhat larger reading would be obtained than the actual diameter. Improper usage in this manner could eventually lead to the conditions of II-C, whereby the movable jaw becomes sprung, or wear between the jaw and the graduated bar is excessive. It may be noted that wear from extensive general use could produce similar conditions, and checking the zero setting as illustrated in C or setting measuring surfaces of the instrument in this position may show these factors quite distinctly. 47

RULE OR
BAR

ADJUSTING SCREW

CARRIER

LOCK SCREW

ADJUSTING NUT

GIB

VERNIER PLATE

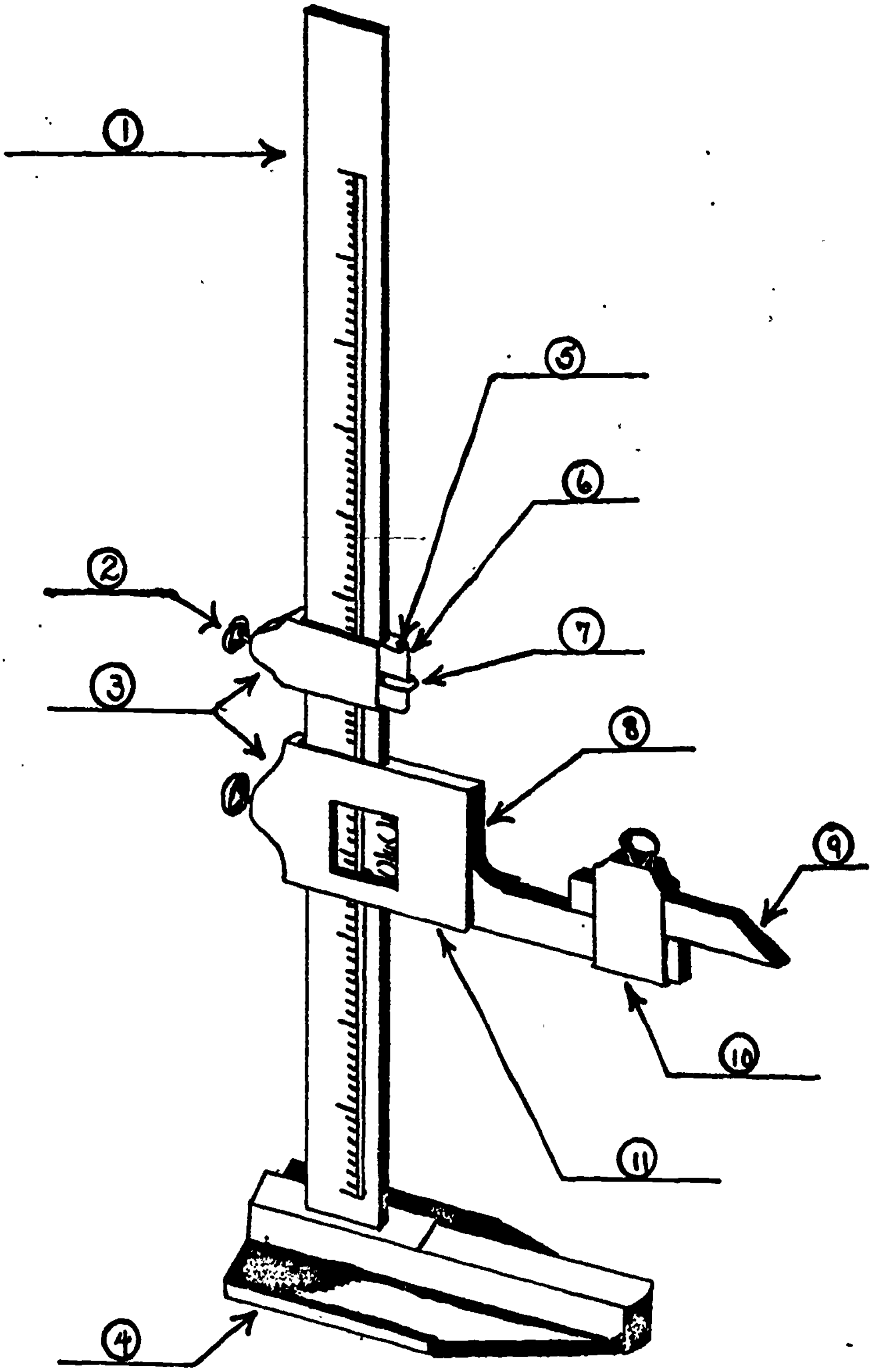
SCRIBER

SCRIBER
CARRIER

ADJUSTING JAW

FIXED JAW
OR BASE

VERNIER
HEIGHT GAGE


1
2
3
4
5
6
7
8
9
10
11

$$25\overline{)\,.600\,}\quad = .024$$

$$\begin{array}{r} .024 \\ 25\,)\overline{.600} \\ \underline{50} \\ 100 \\ \underline{100} \end{array}$$

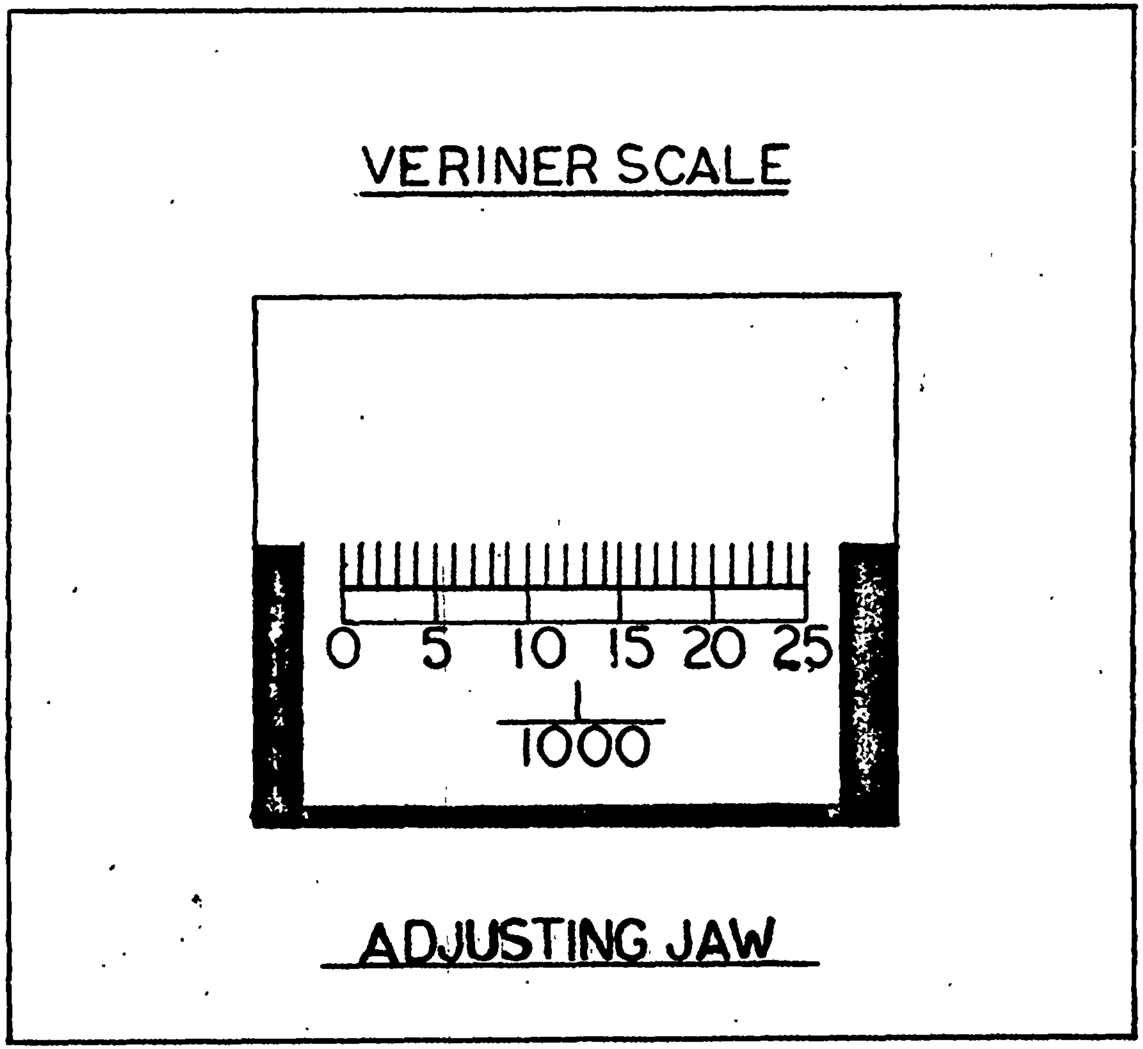
VERNIER SCALE .600 LONG GRADUATED INTO 25 EQ. PARTS

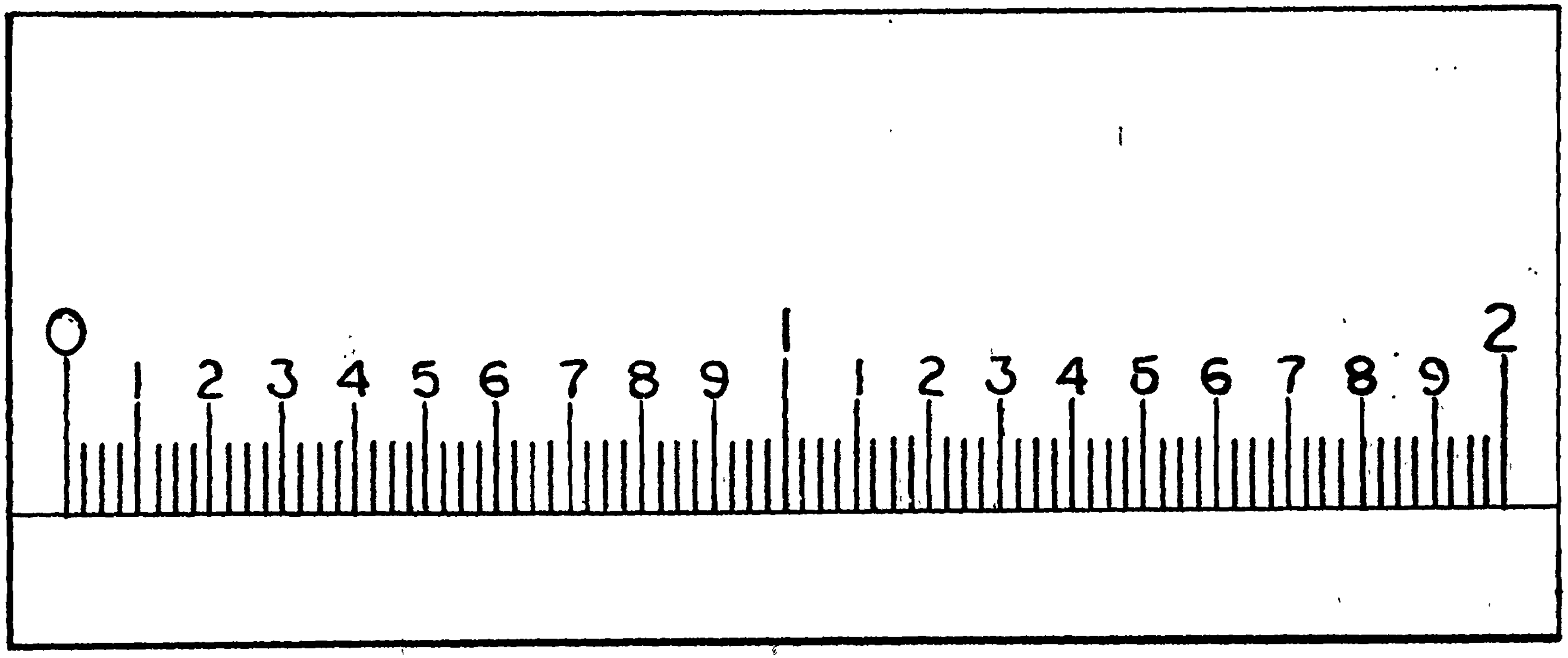
$$\begin{array}{r} .025 \\ \underline{\times\ 24} \\ 100 \\ \underline{50\ \ } \\ .600 \end{array}$$

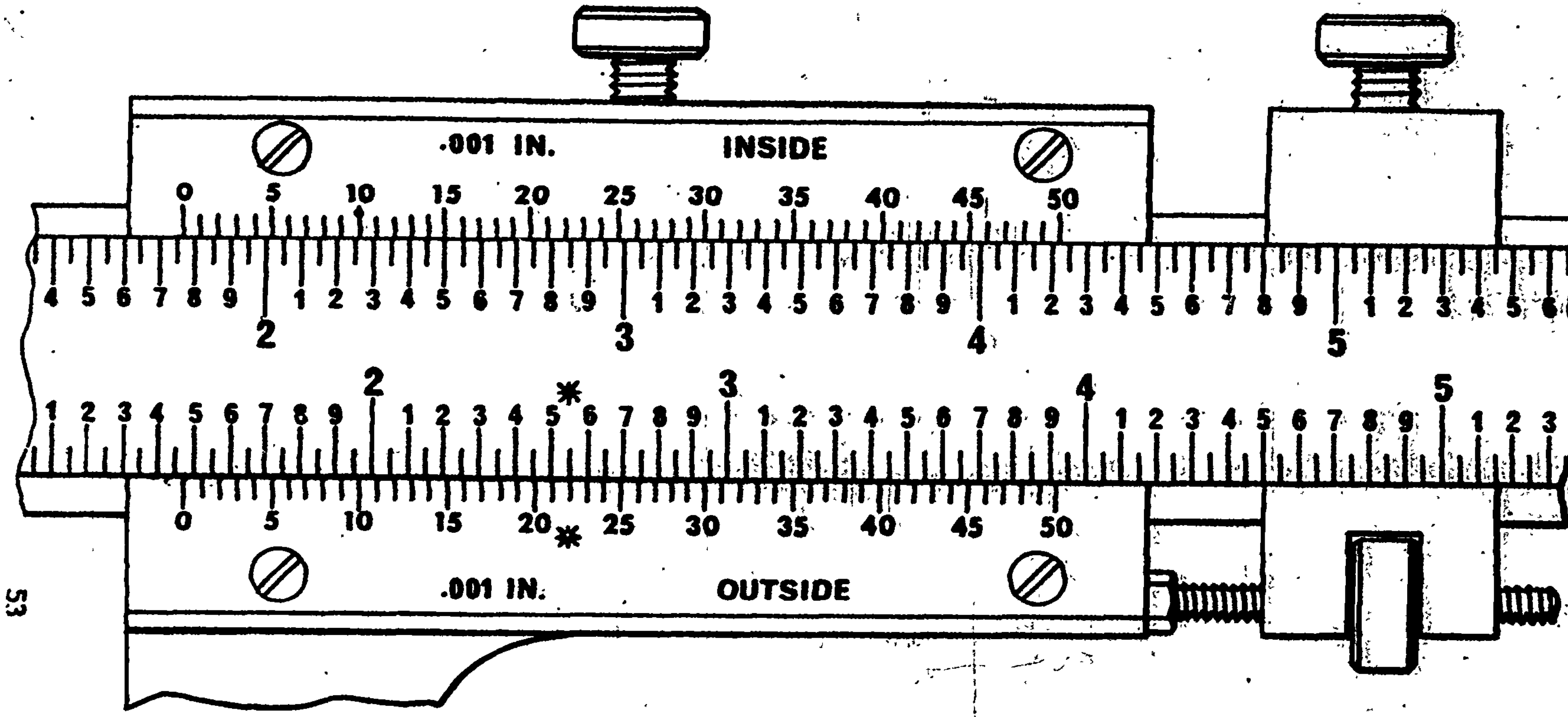
RULE GRADUATIONS ARE .025 SO IN .600 WE HAVE 24 EQ. PARTS

THIS MEANS THAT EACH GRADUATION ON THE RULE AND SCALE DIFFER .001 OF AN INCH SO THAT ONLY ONE LINE ON THE SCALE AND RULE IN .600 WILL EXACTILY COINCIDE.

WHERE THE TWO LINES COINCIDE IS THE POINT WE READ THE VERNIER SCALE


VERINER SCALE
0 5 10 15 20 25
1
1000
ADJUSTING JAW


0 1 2 3 4 5 6 7 8 9 1 1 2 3 4 5 6 7 8 9 2



VERNIER CALIPER SCALE

UNIVERSAL BEVEL PROTRACTOR

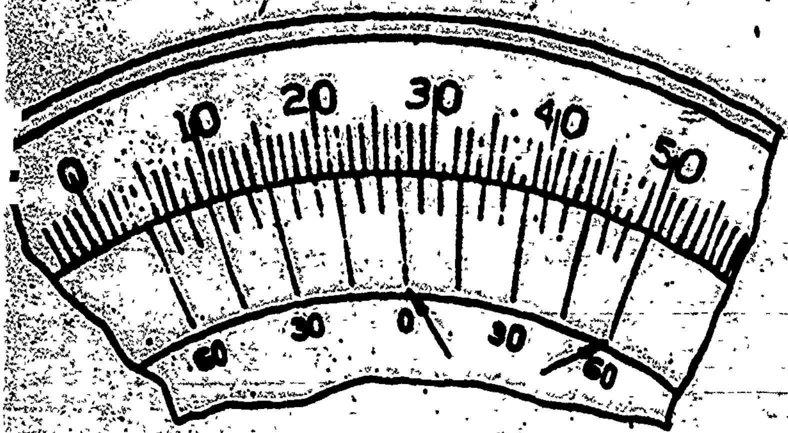


FIGURE I

Figure I illustrates how readings are obtained from the universal bevel protractor. The whole degrees from which the zero on the vernier has moved from zero on the dial are noted (27°). Then to determine the minutes from the vernier scale, one must read in the same direction from zero on the vernier as the vernier moved from zero on the dial (to the right in Figure I). In the illustration, the zero and 60 divisions line up exactly indicating whole degrees.
The reading is then 27° 0'.

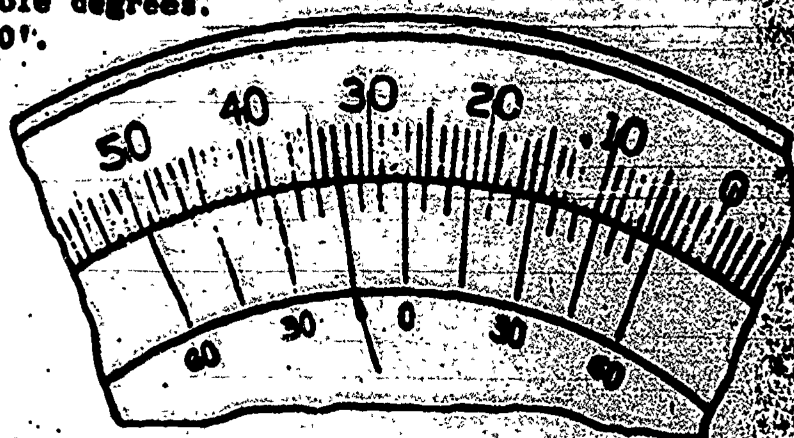
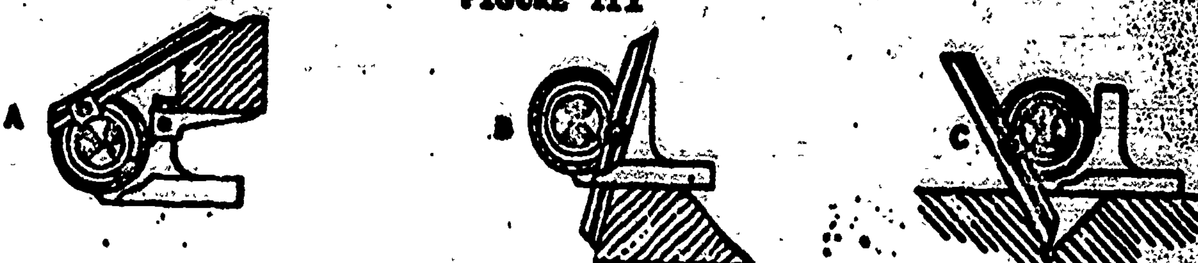


FIGURE II

In Figure II the number of whole degrees are noted (27°). Then by reading the vernier to the left (same direction from zero as the vernier moved from zero on the dial), the number of minutes may be obtained. In the illustration, the line on the vernier which coincides best with a division on the dial is the 15. The reading therefore is 27° 15'.

FIGURE III

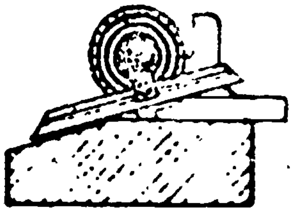


The universal bevel protractor may be used to measure nearly any angle of the features of a work piece.

Figure III A. Use of acute angle attachment - measuring small or acute angles.
B. Obtuse angle measurement - outside features.
C. Obtuse angle measurement - inside features.

USE OF THE UNIVERSAL BEVEL PROTRACTOR

FIGURE I



The illustration in Figure I serves to show improper application of the universal bevel protractor. In this instance, the instrument would produce a reading somewhat greater than the actual angle formed by the two surfaces of the work piece. Conditions such as this could be caused by manipulative errors, burrs, dirt, or poor reference surfaces. These factors would have to be given consideration, to provide for the blade having a good bearing on the surface of the work piece.

In Figure II an incorrect reading would also be obtained as the blade and base of the instrument do not have a true bearing on the work piece. Foreign objects and manipulative errors would also be factors which could cause this condition. The reading obtained would actually be somewhat less than the work piece features, as the blade has not been opened the desired amount to duplicate the angle between the two surfaces.

FIGURE II

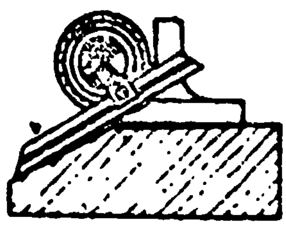
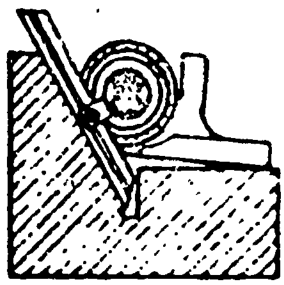


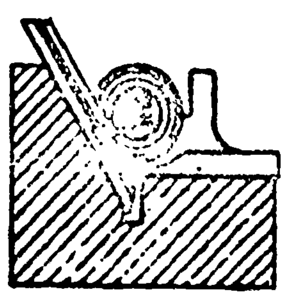
FIGURE III



Another error in application of the vernier bevel protractor is illustrated in Figure III. Here again the blade has not been opened a sufficient amount, relative to the base of the instrument. This illustrates very distinctly how an obstruction upon the surface of the work could cause errors. In the illustration the base of the instrument is not bearing on the surface of the work. This might also be caused by applying too much force upon the blade and causing it to ride too far up along the surface of the work.

Proper application of the universal bevel protractor is shown in Figure IV. In this example the blade and the base of the instrument have proper bearing upon the surfaces of the work piece. This is the basis of measurement of the universal bevel protractor, whereby the instrument reproduces or duplicates the features of the work. With the part feature properly duplicated, the value of the angle in degrees and minutes may be read from the instrument.

FIGURE IV



SNAP GAGE APPLICATIONS

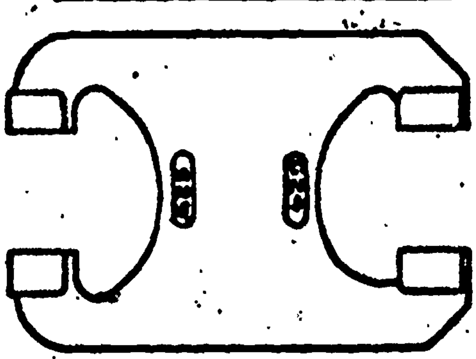
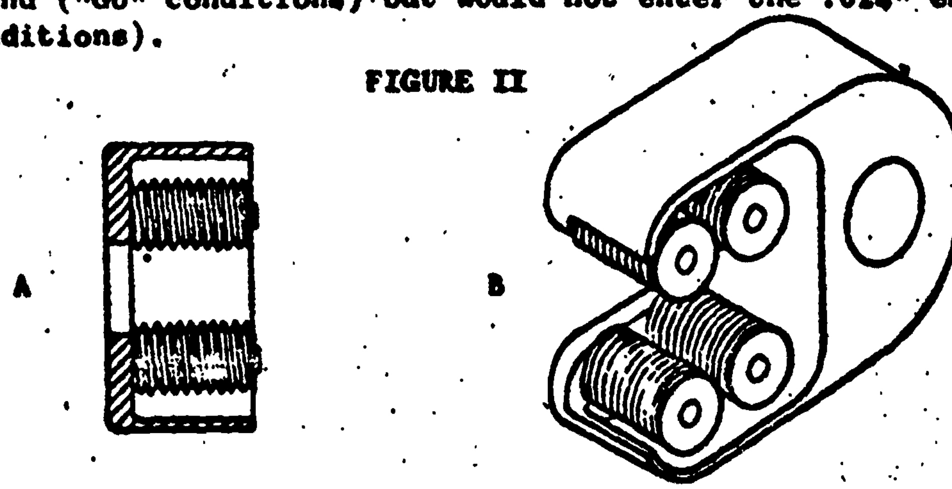


FIGURE I

The illustration above shows the features of a standard, solid-construction, 5/8" limit snap gage. The basic design of this gage is somewhat similar to that of all snap gages, that is the U-shaped frame with accurately machined blocks attached and set at specific dimensions. These may also be adjustable and are used primarily for gaging external surfaces or diameters. A work piece within the tolerance of this gage would enter the .625" end ("Go" conditions) but would not enter the .624" end ("No Go" conditions).

FIGURE II

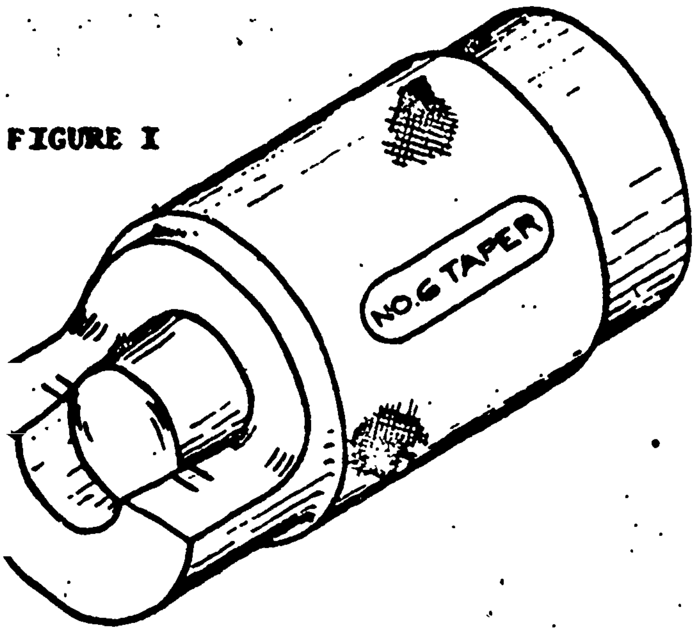


The threaded gage as illustrated in Figure II is used to gage the limits of tolerance of the major, minor, and pitch diameters of an external thread. They are designed for threads of a particular pitch and class. Variation in class of threads will also vary the limits of tolerance of the pitch diameter. The two sets of rolls attached to the frame determine whether the work piece is within tolerance. The threaded work piece will engage readily with the rolls as can be seen with end view A. If the work piece is within tolerance, it will pass through the initial set of rolls with no more than a slight drag ("Go"), but will catch in the second set ("No Go"). The rolls will revolve when engaged, thereby distributing wear evenly around the periphery of the rolls. These gages are produced in a large variety of designs according to varying size, pitch, class, and thread form.

TAPER RING GAGES

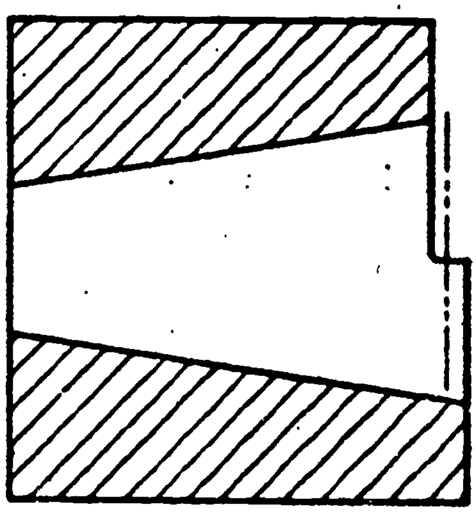
BEST COPY AVAILABLE

FIGURE I



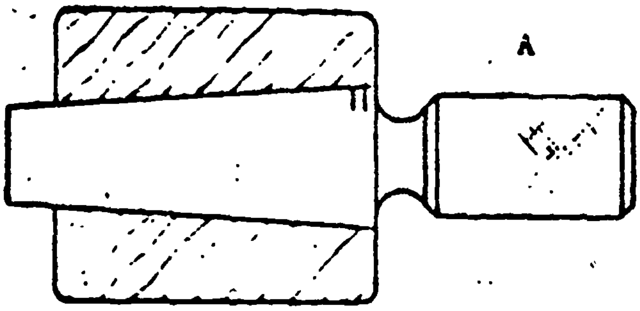
In the illustration at the left, an application of the taper ring gage is shown. The work piece inserted in the gage is within the limits of tolerance as shown by the two scribed lines on the gage. The number 6 on the gage may indicate different items according to the type of taper. A number 6 Brown & Sharpe would have approximately .500" taper per foot with a diameter at the small end of .500" and a length of 2-3/8". The number 6 Jarno has a taper of .600"/foot with a small diameter of .600" and a length of 3". The Morse number 6 has a taper of approximately .625" /foot with a small diameter of 2.116" and a length of 7-3/8".

FIGURE II

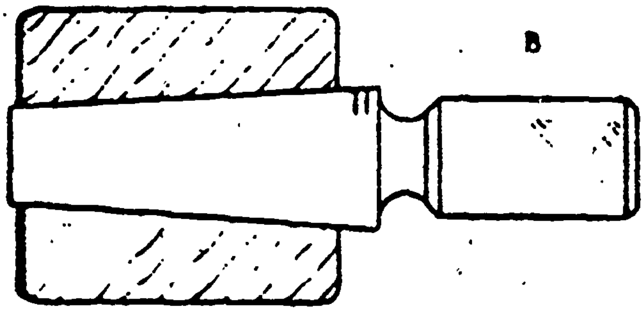


The taper ring illustrated in Figure II serves to illustrate variation in design of some of the taper ring gages. This could be either of the particular types of numbers, but the basic difference arises out of the steps on the end of the gage indicating the limits of tolerance. The ideal position for the end of the tapered work piece would be along the broken line in the illustration, i.e., exactly between the limits of tolerance.

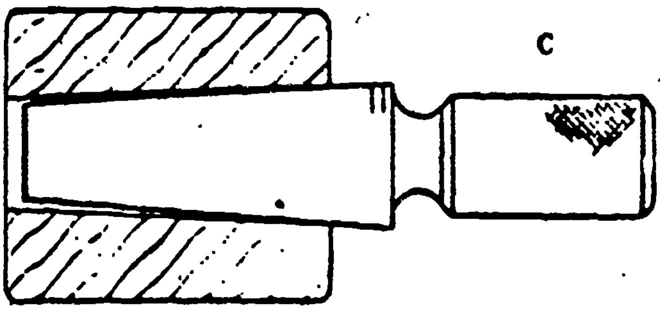
APPLICATION OF THE TAPER PLUG GAGE



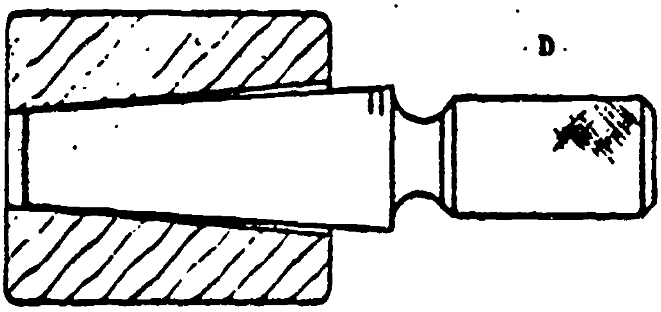
A



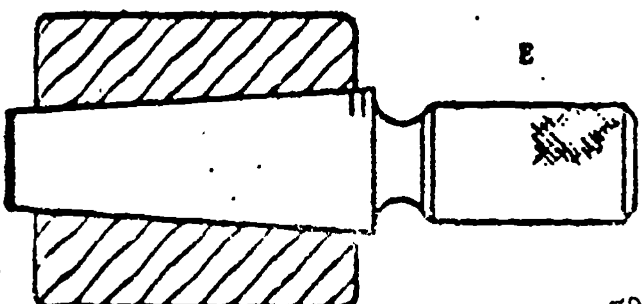
B



C



D



E

The drawings at the left illustrate a number of errors which may occur when producing tapered work pieces. They also serve to show how these errors are detected through application of the taper plug gage. Detecting errors in the diameter size is determined by the depth of insertion of the plug gage. With A and B the amount of taper is correct, but too much stock has been removed - diameters too large in A, and not enough stock has been removed - diameters too small in B.

Incorrect taper is illustrated in C and D; taper in C not pronounced enough - too little taper per foot, and taper in D too pronounced - too much taper per foot!

One must keep in mind that tapers may vary with different types such as Brown & Sharpe, Jarno, and Morse, and within these types there are various numbers which may vary slightly in taper per foot and diameter sizes. It is important that the correct number and correct type of taper are chosen for a particular application. Once this is achieved and the correct taper is machined, the conitions in E may be attained.

With E the taper is correct as illustrated by the bearing between the work and plug gage, and the diameter tolerance is maintained as shown with the end of the work falling between the two scribed lines on the plug indicating the limits of size. Important considerations here are dirt, burrs, and heat as well as several other items which would greatly hinder precision and reliability.

THREAD GAGE DATA

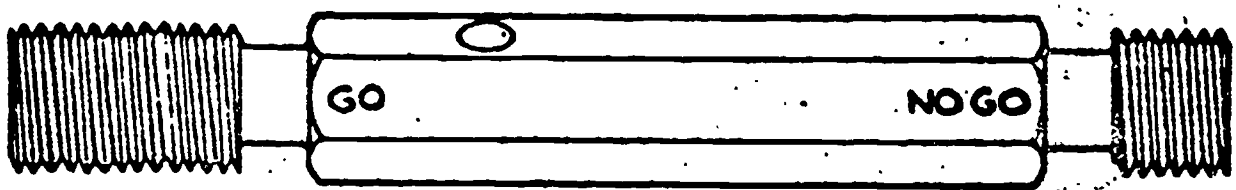
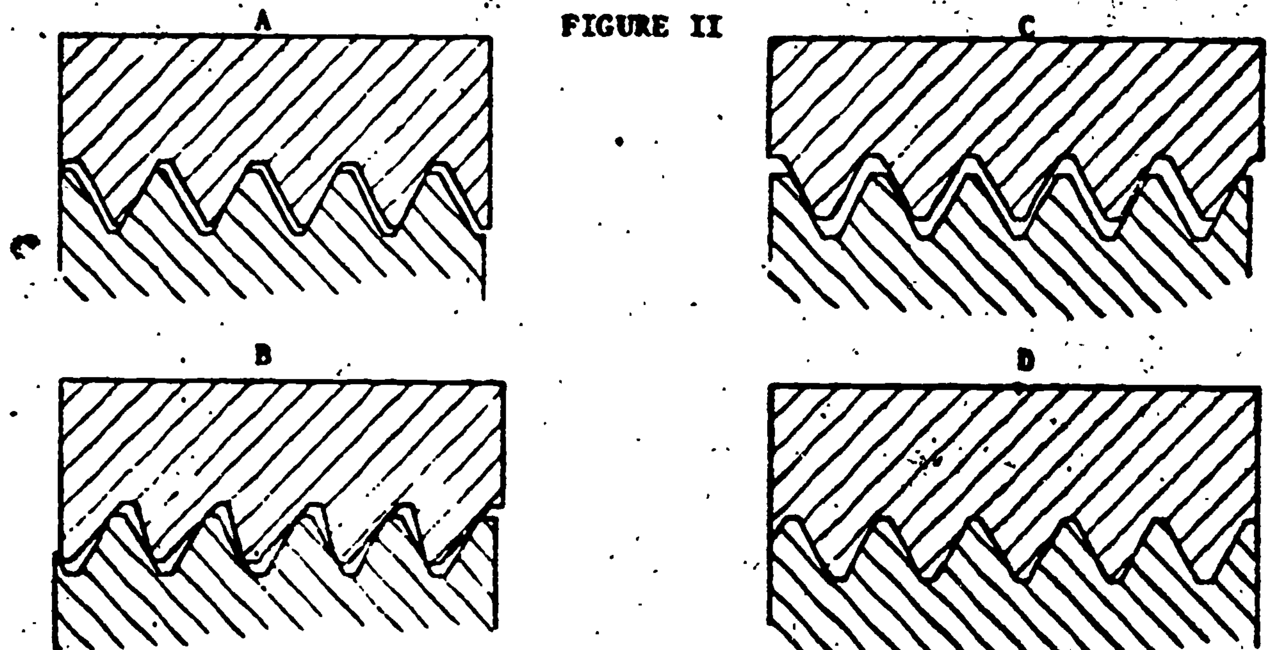


FIGURE I

FIGURE II



The figures above illustrate some of the various features of the thread plug gage and the mating of threaded work with the thread gage. In Figure I a design of the thread plug gage is illustrated quite effectively. This gage checks the major, minor, and pitch diameters of the thread: the "Go" member checks the major diameter of the smallest threaded hole, and the "No Go" member checks the pitch diameter; must not enter more than two turns. With Figure II, the top portions of the illustrations represent the actual work piece with the bottom portions being the thread plug gage. In A the diameters are cut too large (thread too deep), and there would be play between the work and the gage. A similar condition exists with B. More important however, is the fact that the pitch or lead varies and is incorrect along the length of the thread. In C the helix angle of the thread is incorrect and though (same as in B) the feel of the plug in the hole might seem correct, it is easy to see how these threads would soon peen down, thereby producing an undesirable condition. This illustrates the importance of sharp and true cutting tools. The conditions in D illustrate the fit between threads one must always strive for. There is a relatively perfect fit between the mating parts.

DIAL INDICATOR FEATURES

FIGURE I

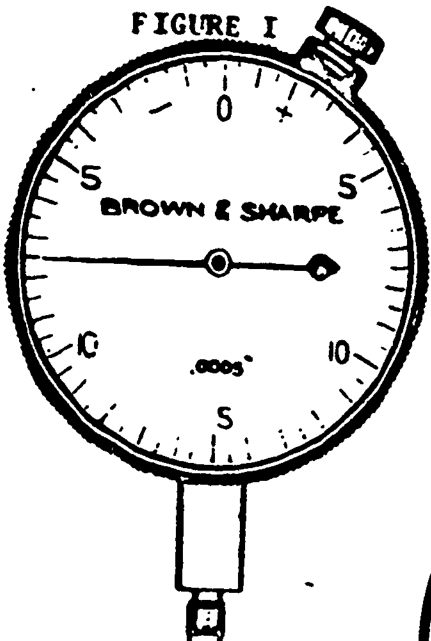


FIGURE II

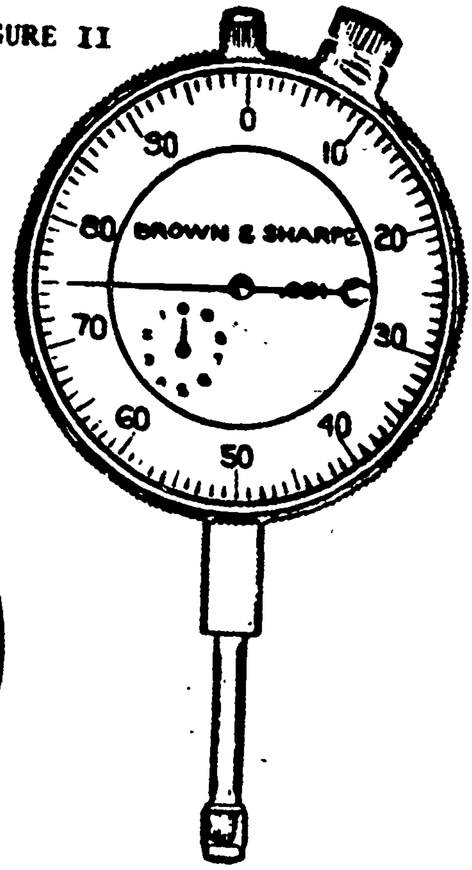
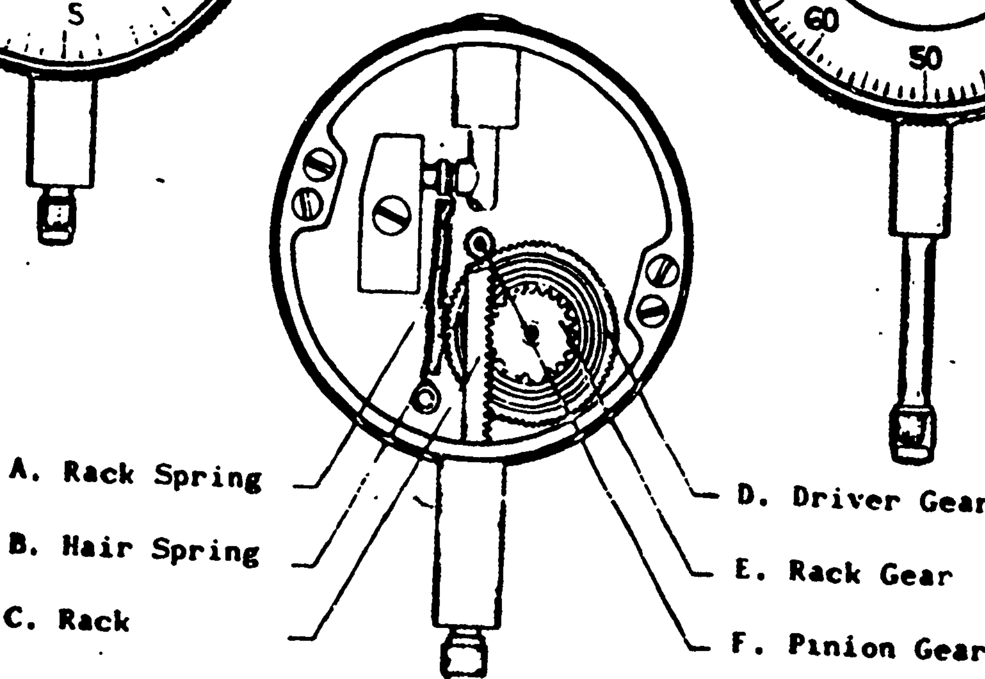


FIGURE III

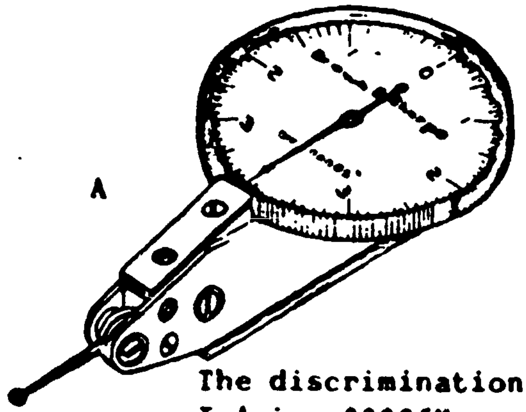


ne general features of the dial indicator instruments are illustrated in the igures above. Figures I and II serve to show a variety of ranges, scale raduations, and different types of dials. Figure I illustrates the balanced ial with a range of .030" and .0005" scale divisions. The indicating hand at rest at the nine o'clock position as with all indicators, and the full avel would be 2 1/2 revolutions of the hand. Zero setting is possible hrough unlocking the bezel screw at the upper right hand position of the in- icator and rotating the dial. The long range dial indicator with the contin- dial is illustrated in Figure II. The range of the dial is .100", and dial is graduated in .001" increments. The total range of the instrument one inch as it has a supplementary dial to count the revolutions. Ampli- ication is achieved with these indicators through the system of gears or a omewhat similar gear set-up to that illustrated in Figure III. As the spin- e is moved, the rack in turn rotates the rack gear which is mounted on the shaft as the driver gear. This large driver rotates the pinion gear which connected directly to the indicating hand and this is how readings are ob- ained. The ratio of these gears is the amount of amplification. The hair ing acts to take up any backlash in the gear assembly, and the rack spring ts to bring the instrument back to the rest position and maintain gaging ressure.

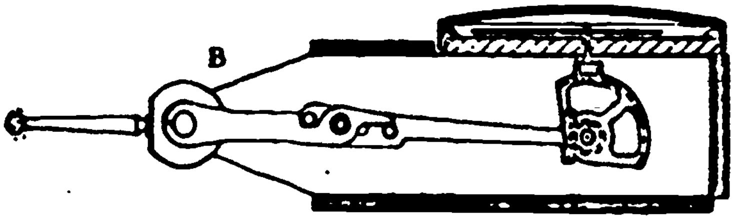
THE TEST INDICATOR AND DIAL INDICATOR USAGES

FIGURE I

A



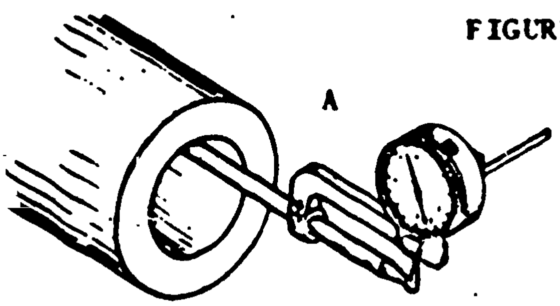
B



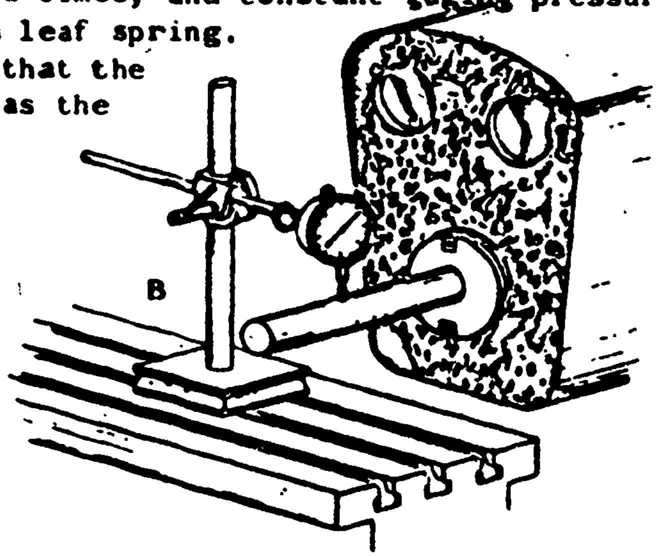
The discrimination of the test indicator illustrated in Figure I-A is .00005", and the range of the dial is .005". This instrument is generally more accurate and more sensitive than the ordinary dial indicator. These instruments duplicate measuring pressure for numerous readings, and they provide for precision transfer and comparison measurement. The high level of discrimination is achieved through the combined ratios of the lever and gear mechanisms illustrated in Figure I-B. Amplification is well over five hundred times, and constant gaging pressure is maintained through application of a leaf spring. The design of this instrument is such that the hand rotates in a clockwise direction as the pointer is activated up or down.

FIGURE II

A



B



The dial indicator is used frequently to test the roundness and concentricity features with internal and external applications. The dial indicator may be furnished with a hole attachment to test internal and various other surfaces which cannot be reached with the spindles of dial indicators. Figure II-A illustrates a dial indicator adapted in this manner. The pointer bears on the bore of the cylindrical work piece and tests the concentricity of the work piece relative to the mounting fixture. The work is mounted in a chuck or fixture, and as it revolves, the amount of runout is transmitted and indicated on the dial. This is a particularly useful application for setting up machines prior to production operations. Checking the runout of external diameters of cylindrical work pieces may also be performed as illustrated in Figure II-B. This could be performed to check the roundness or straightness of a milling machine arbor prior to production runs. An application such as this might also be required in a final assembly operation of a milling machine. A test bar would be indicated to gage the various features of the tapered bore of the arbor mounting hole.

TITLE: DRILLS AND DRILLING PROCESSES

UNIT: DRILLS AND DRILLING

OCCUPATION: MACHINIST

OBJECTIVE: 1. To acquaint the student with the types and operations of drill presses.

2. To acquaint the student with drills, reamers, counter sinks, counter bores and other tools used in drill presses and other machine tools.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 4, pages 100-125.

DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. What are the two most common types of drill shanks?
2. How are drill sizes denoted?
3. What is the point angle of a drill?
4. What is the clearance angle of a drill?
5. What is a drill drift?
6. What is a drill Sleeve?
7. How is the speed of a drill calculated?
8. What type of material is used to make drills?
9. How much material is left for reaming?
10. What are the two reasons for reaming a hole?
11. List and give the use of the different types of reamers.
12. Describe the operation of countersinking.
13. Describe the operation of counterboring.
14. What is spot facing?
15. What type of taper is used on taper shanked drills and reamers?

16. How is the size of a drill press determined?

17. List and describe the features of a radial drill press.

18. What are the principal parts of a drill press?

19. What type of chuck is used on most bench drill presses?

20. Why are the points of all drills not ground the same?

FLUTES

HELIX OR RAKE AN

LAND OR MARGIN

HANK

NECK

AXIS

FLUTE LENGTH

HANK

BODY LENGTH

OVER-ALL LENGTH

AND OR MARGIN

CHISEL EDGE

BODY

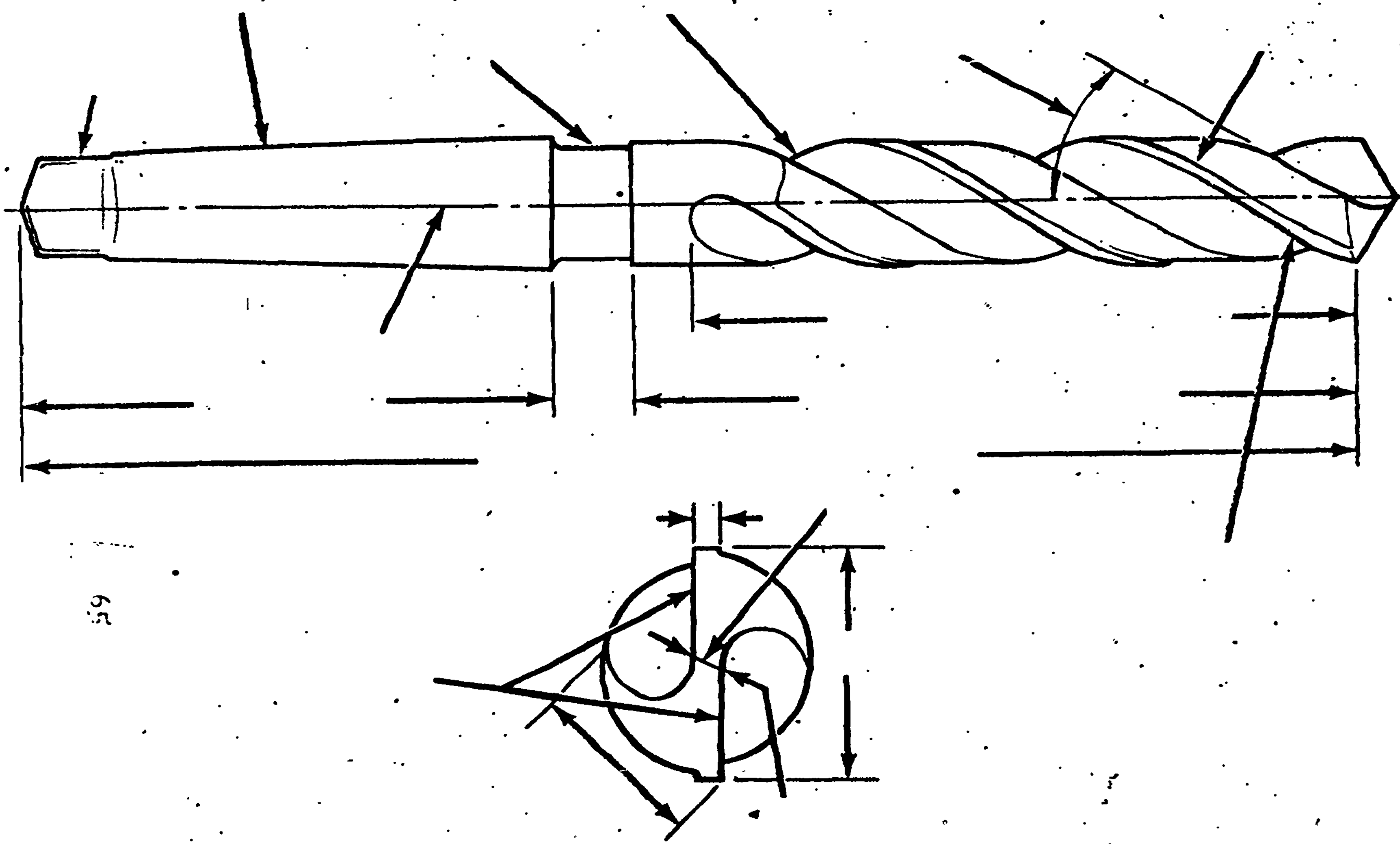
DIAMETER

TTING

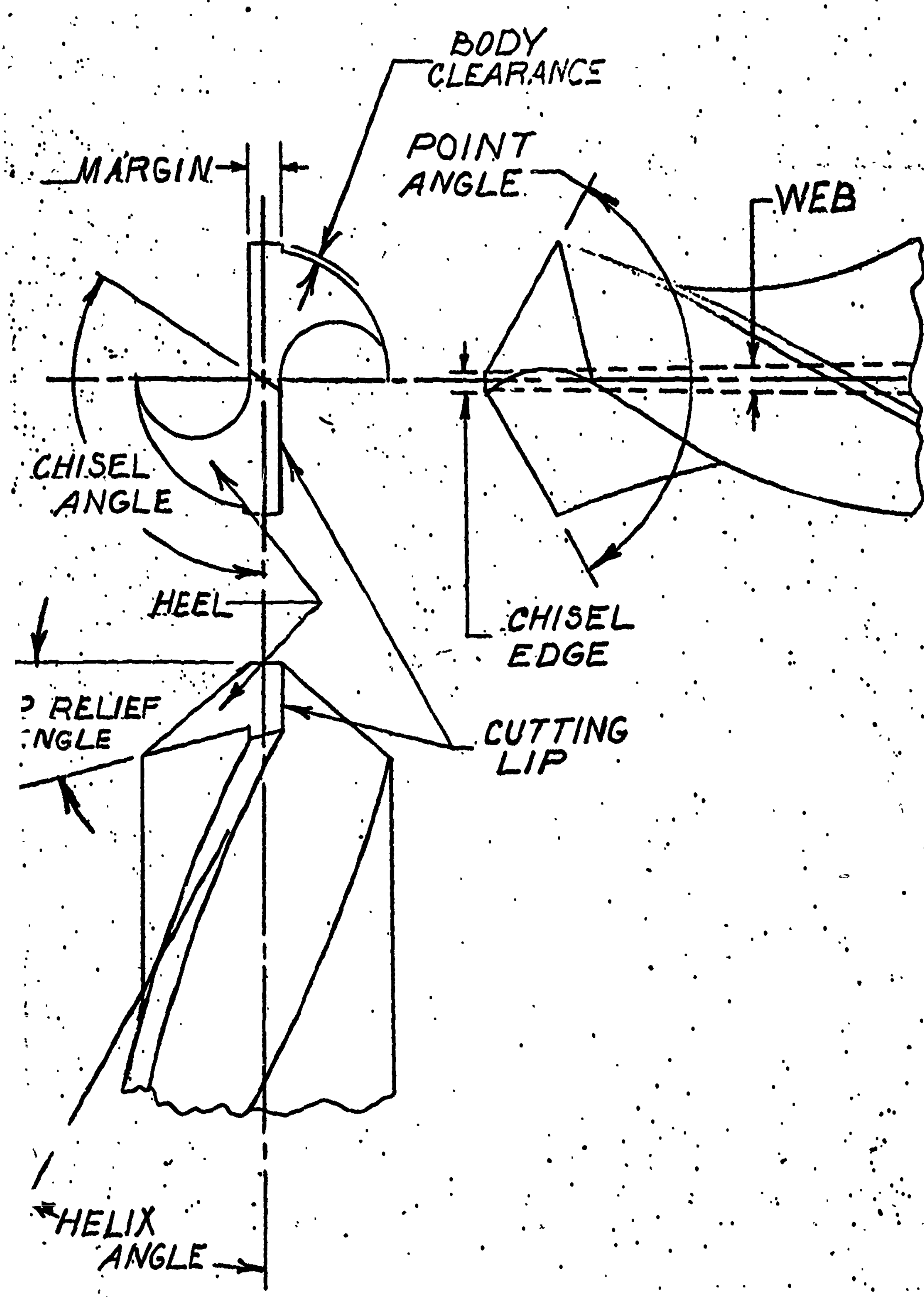
THE POINT

BODY

WEB

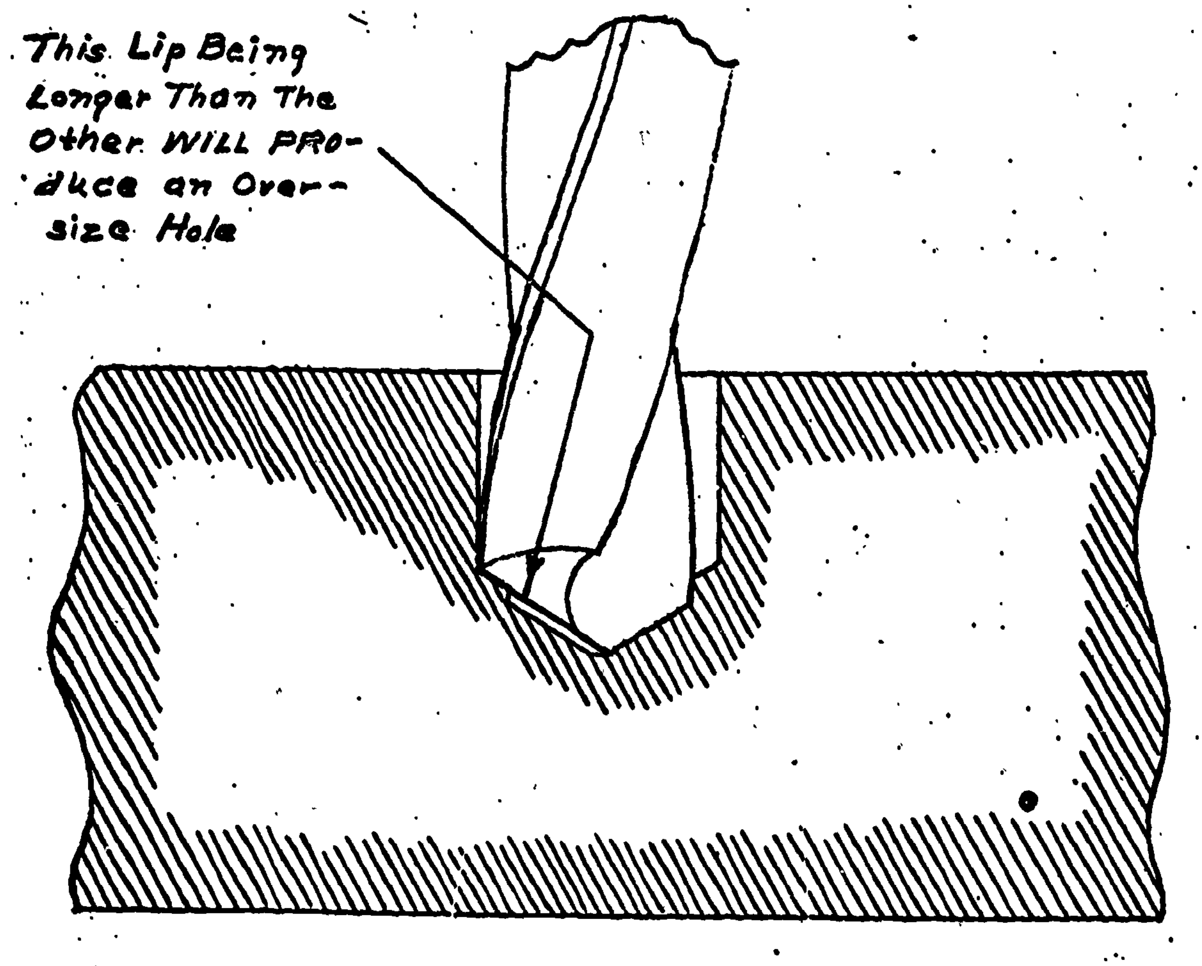


PARTS OF A TWIST DRILL

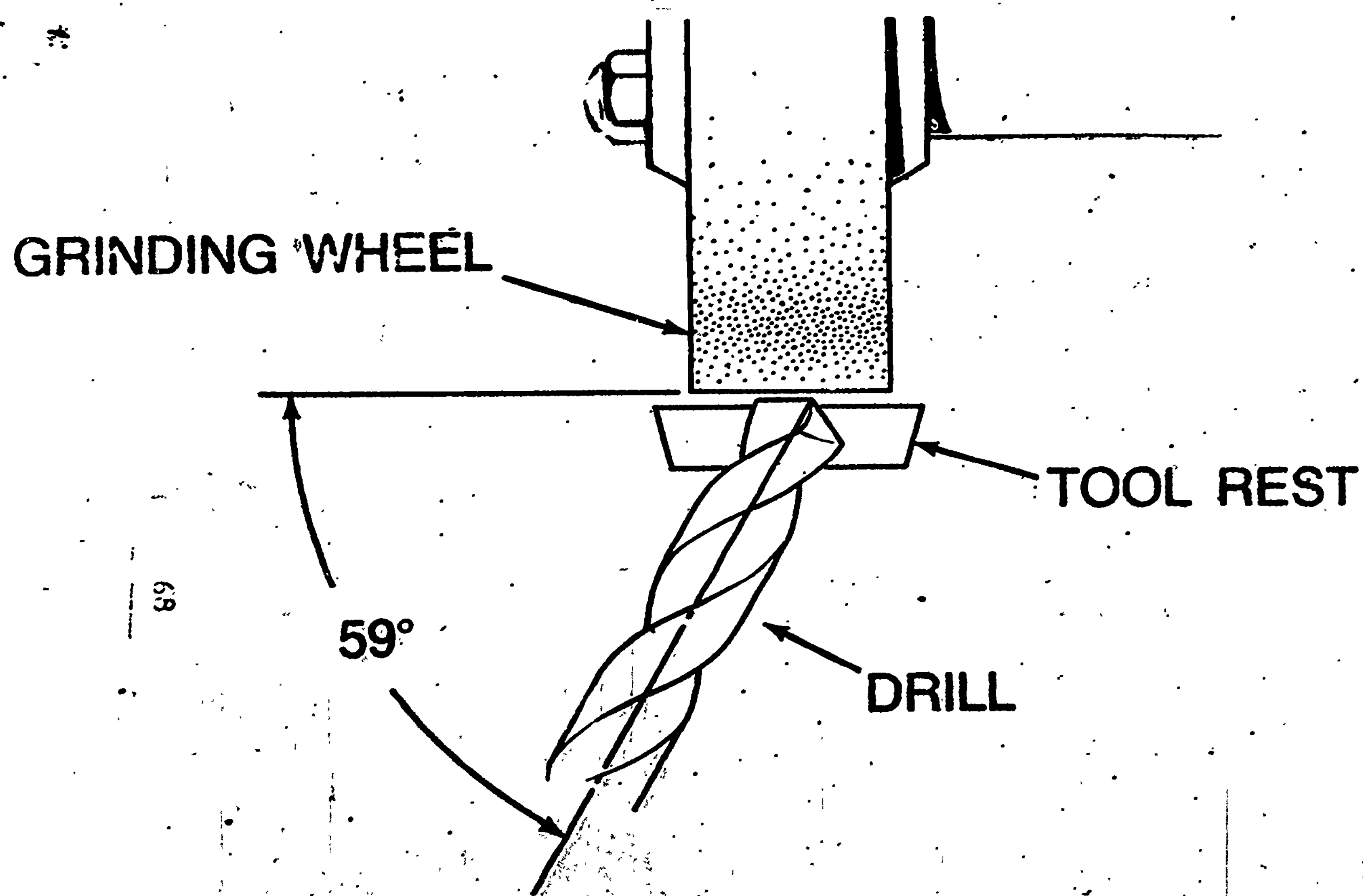


TWIST DRILL ELEMENTS

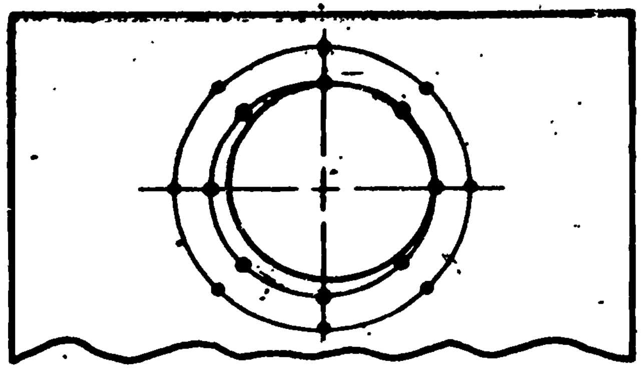
POOR DRILL GRINDING



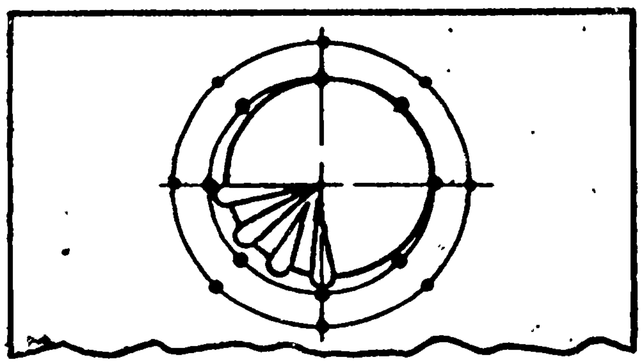
Lips Ground at Same Angles But of Different Lengths Will Produce an Oversize Hole.



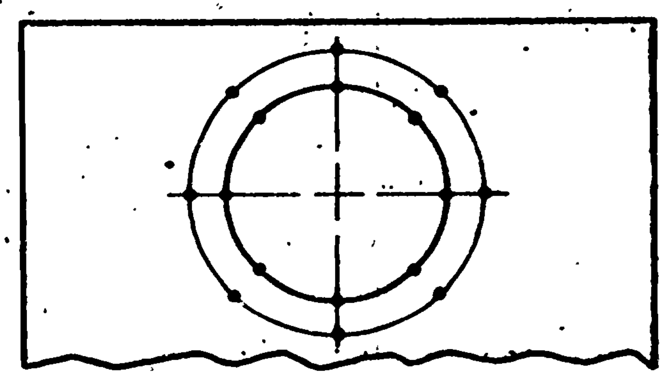
CORRECT ANGLE FOR GRINDING A TWIST DRILL



(a) HOLE DRILLED INCORRECTLY

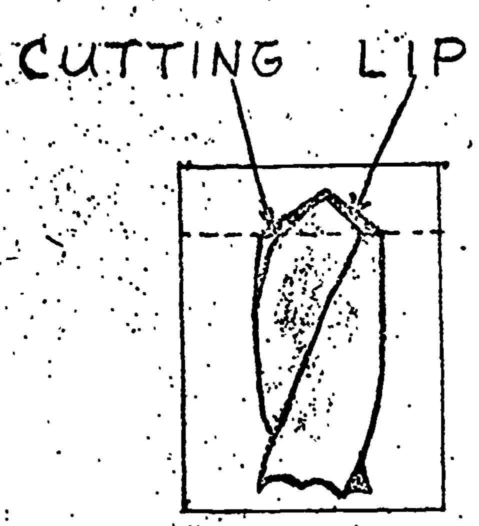


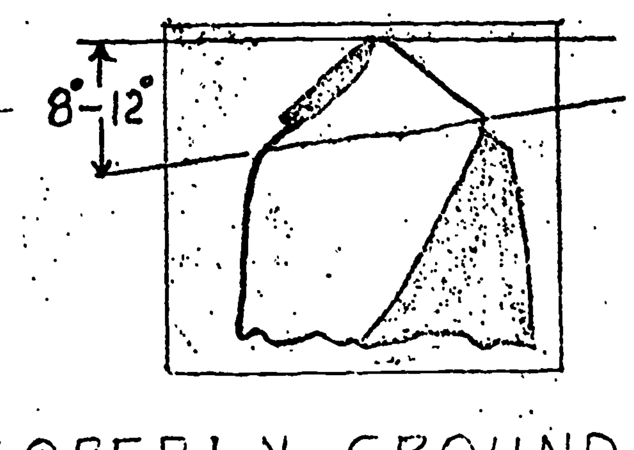
(b) SHALLOW GROOVES CHIPPED ON HEAVY SIDE



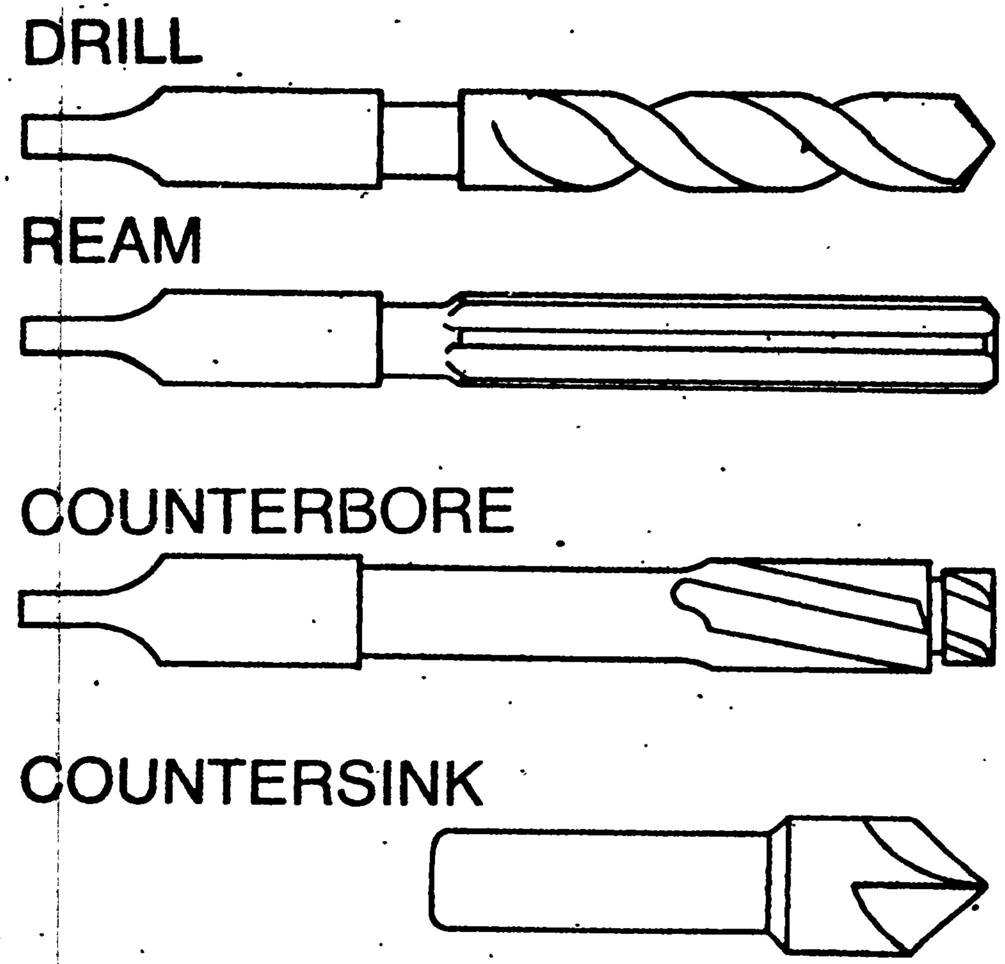
(c) HOLE DRAWN TO CENTRE LOCATION

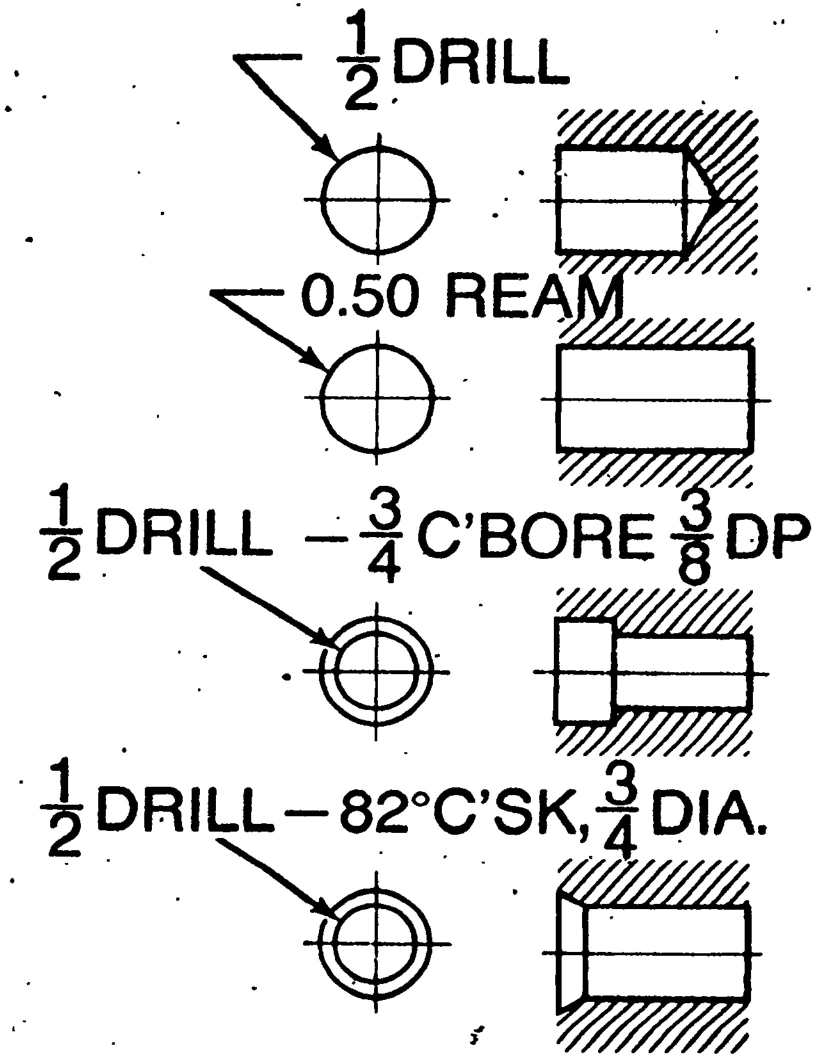
DRAWING A DRILLED HOLE



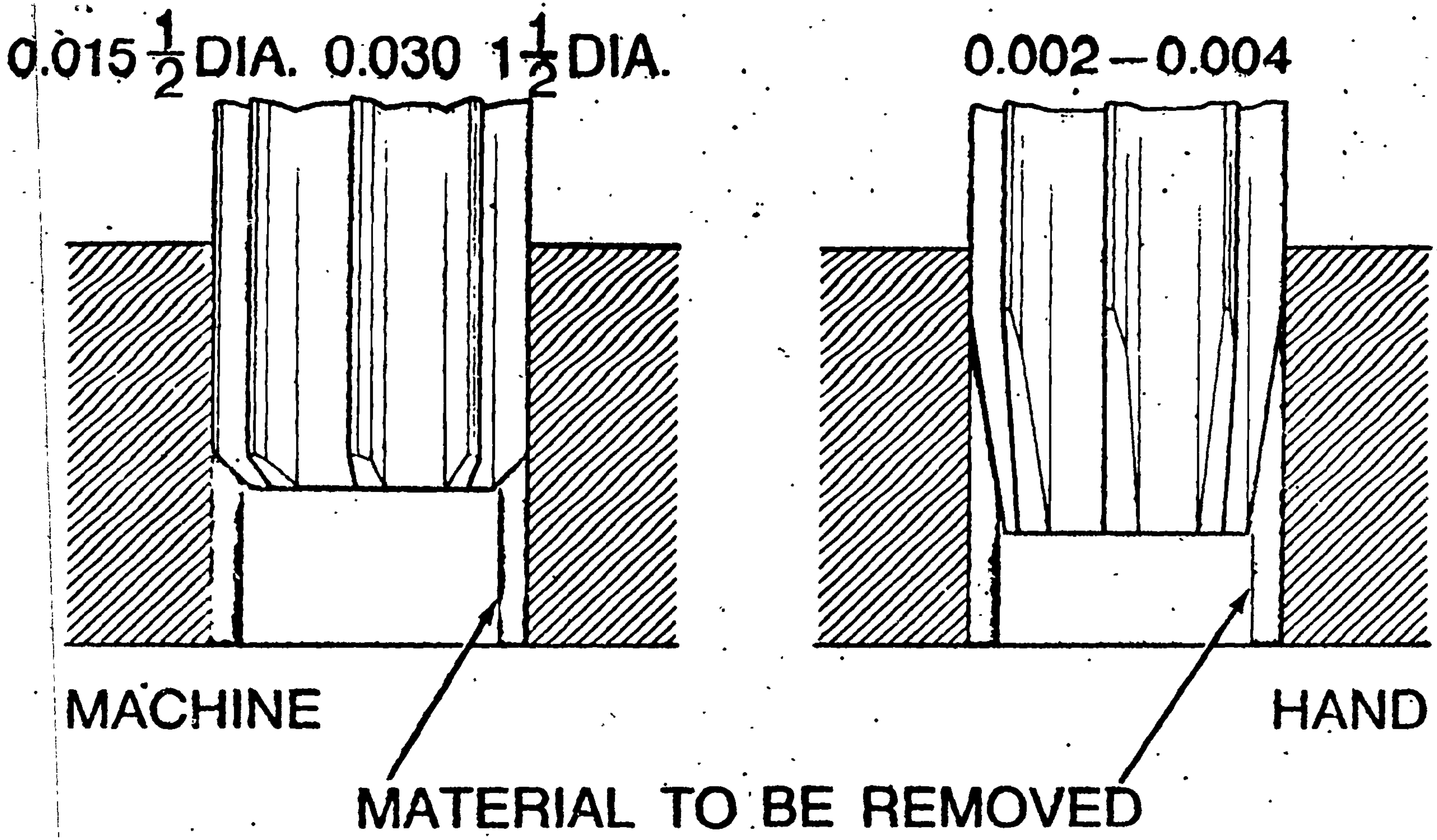


PROPERLY GROUND DRILL


DRILL
REAM
COUNTERBORE
COUNTERSINK

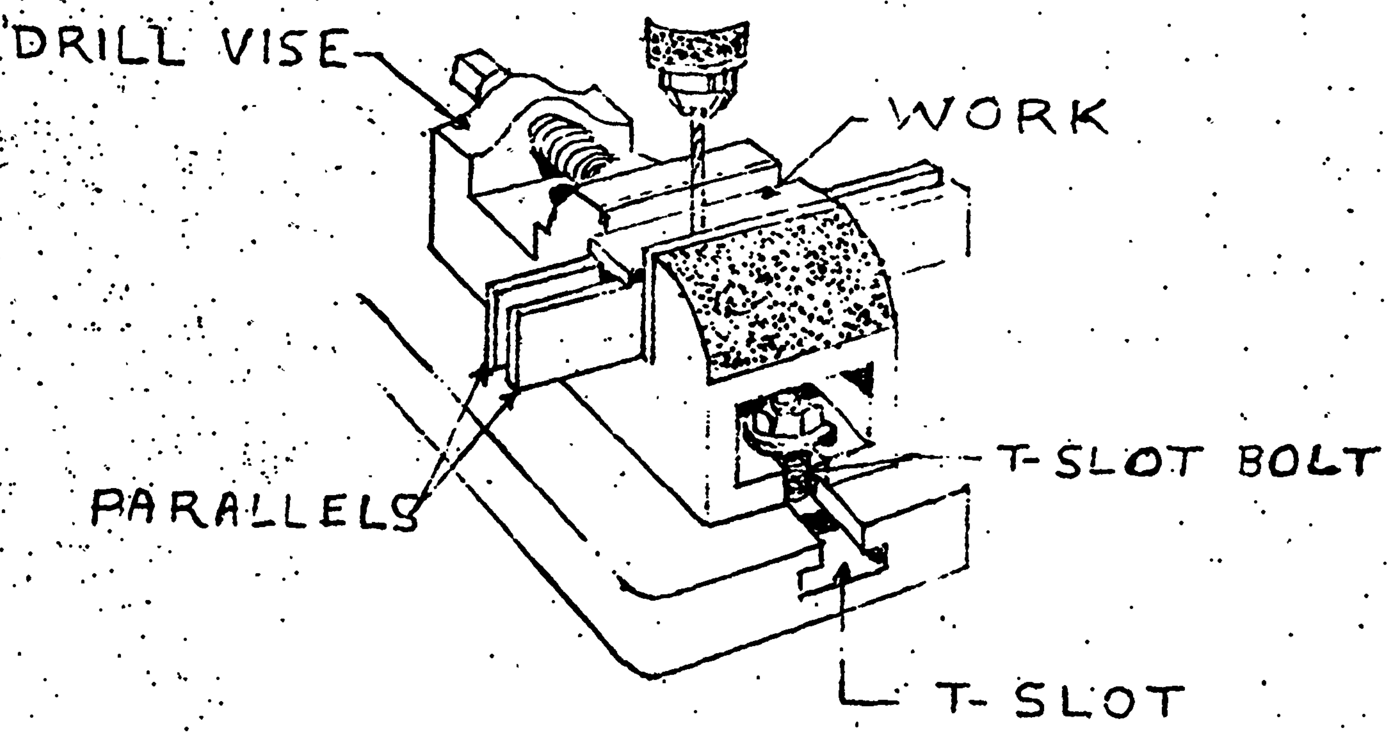


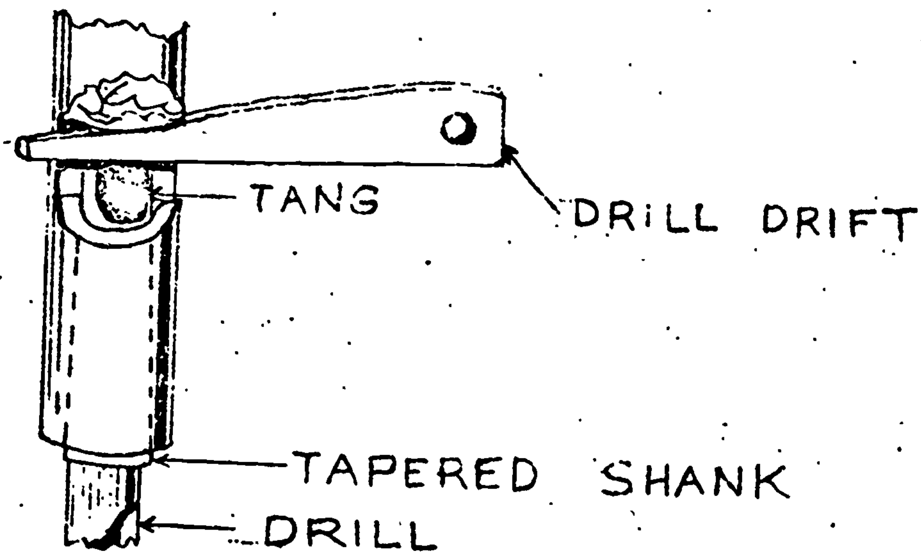
DRILL PRESS OPERATIONS



0.015 $\frac{1}{2}$ DIA. 0.030 1$\frac{1}{2}$ DIA.
0.002—0.004
MACHINE
HAND
MATERIAL TO BE REMOVED

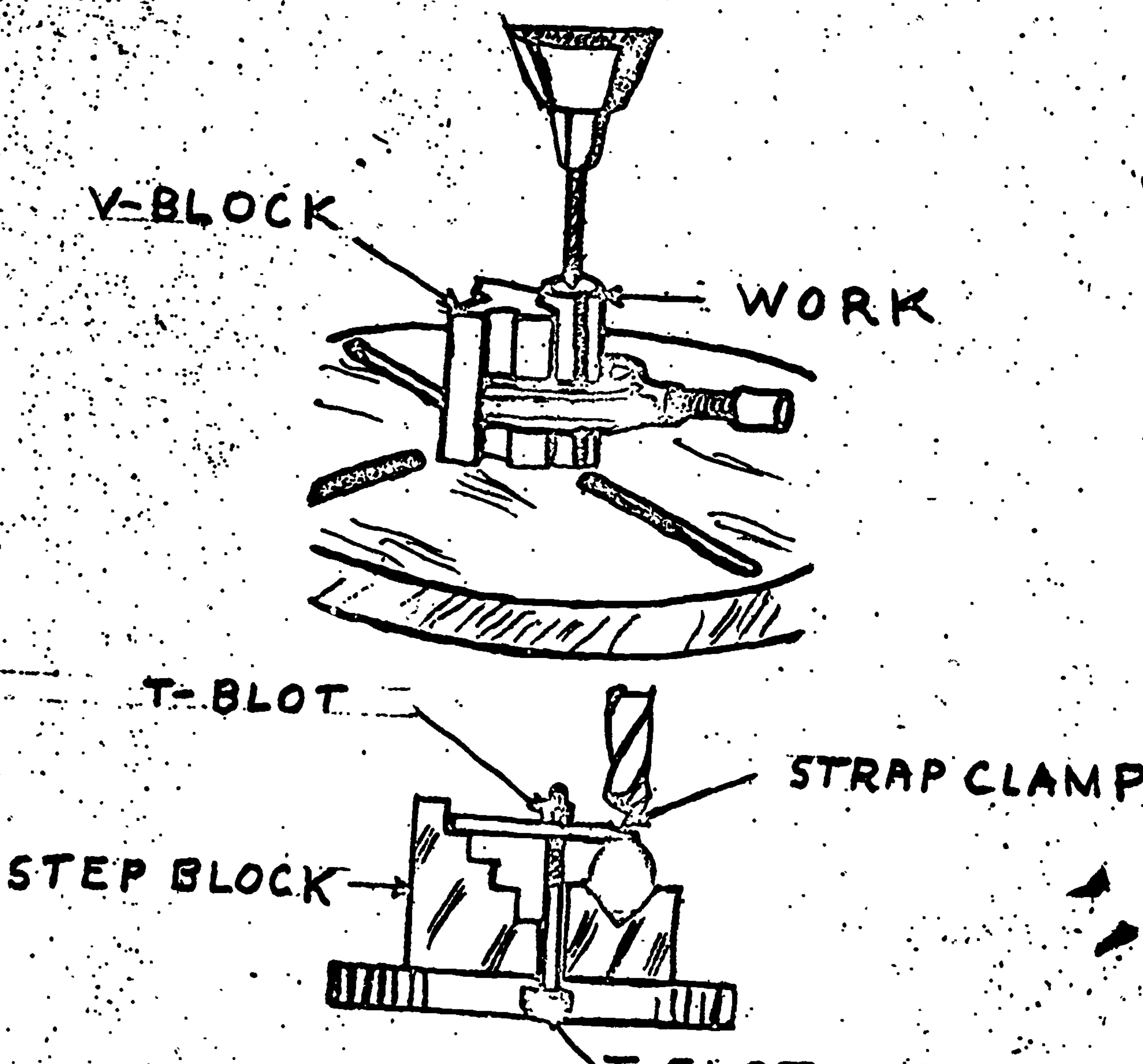
REAMERS


DRILL VISE
WORK
T-SLOT BOLT
PARALLELS
T-SLOT



REMOVING DRILL FROM SPINDLE

NUMBER 1



HOLDING ROUND WORK

TITLE: CUTTING TOOLS

UNIT: SINGLE POINT CUTTING TOOLS

OCCUPATION: MACHINST

OBJECTIVES: 1. To acquant the student with the types of tools used on lathe and shapers.

2. To acquaint the student with the proper cutting and clearance angles of lathe and shaper tools.

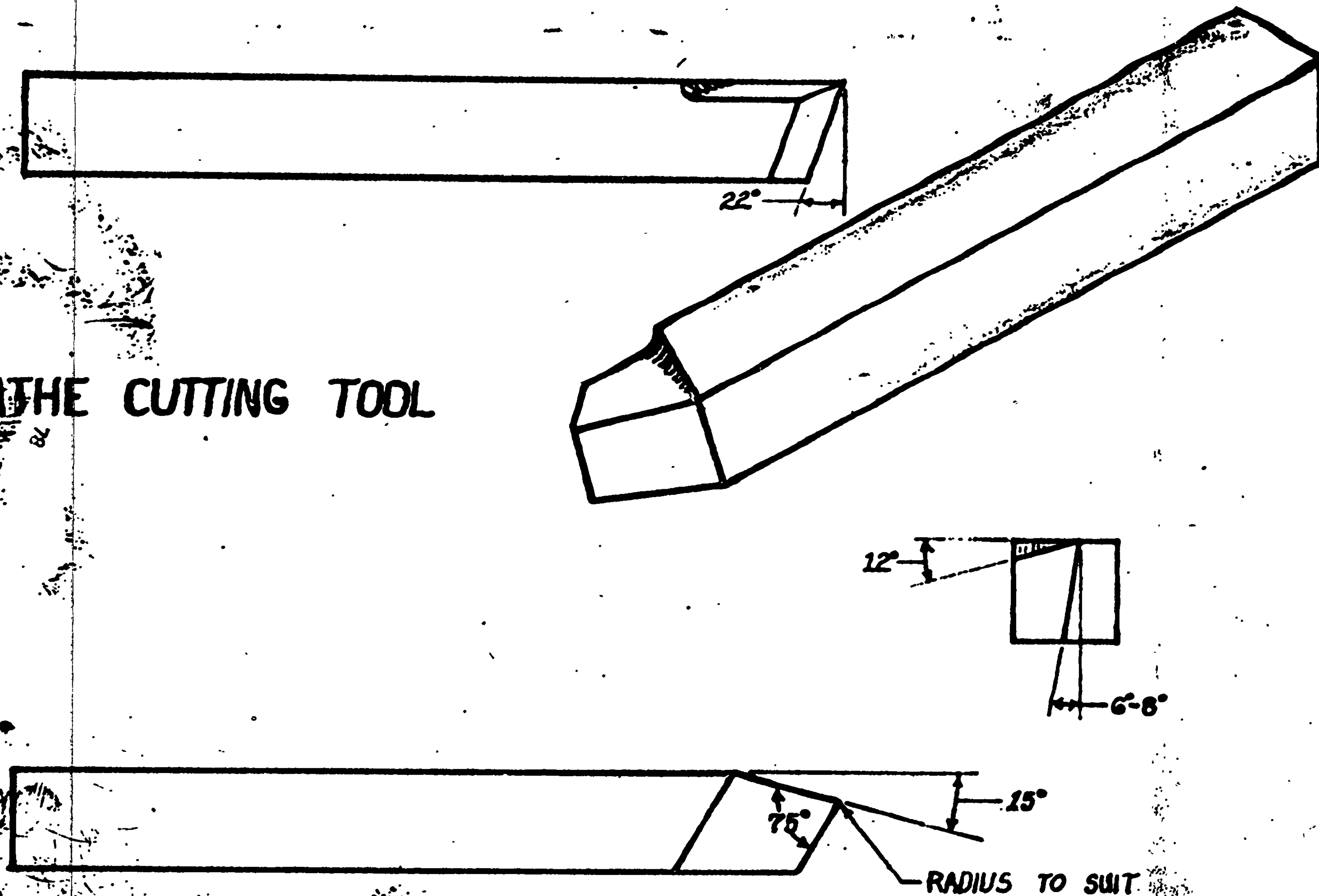
REFERENCES: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 5, pages 126-134

DIRECTIONS: Read the above reference and answer the following questions.

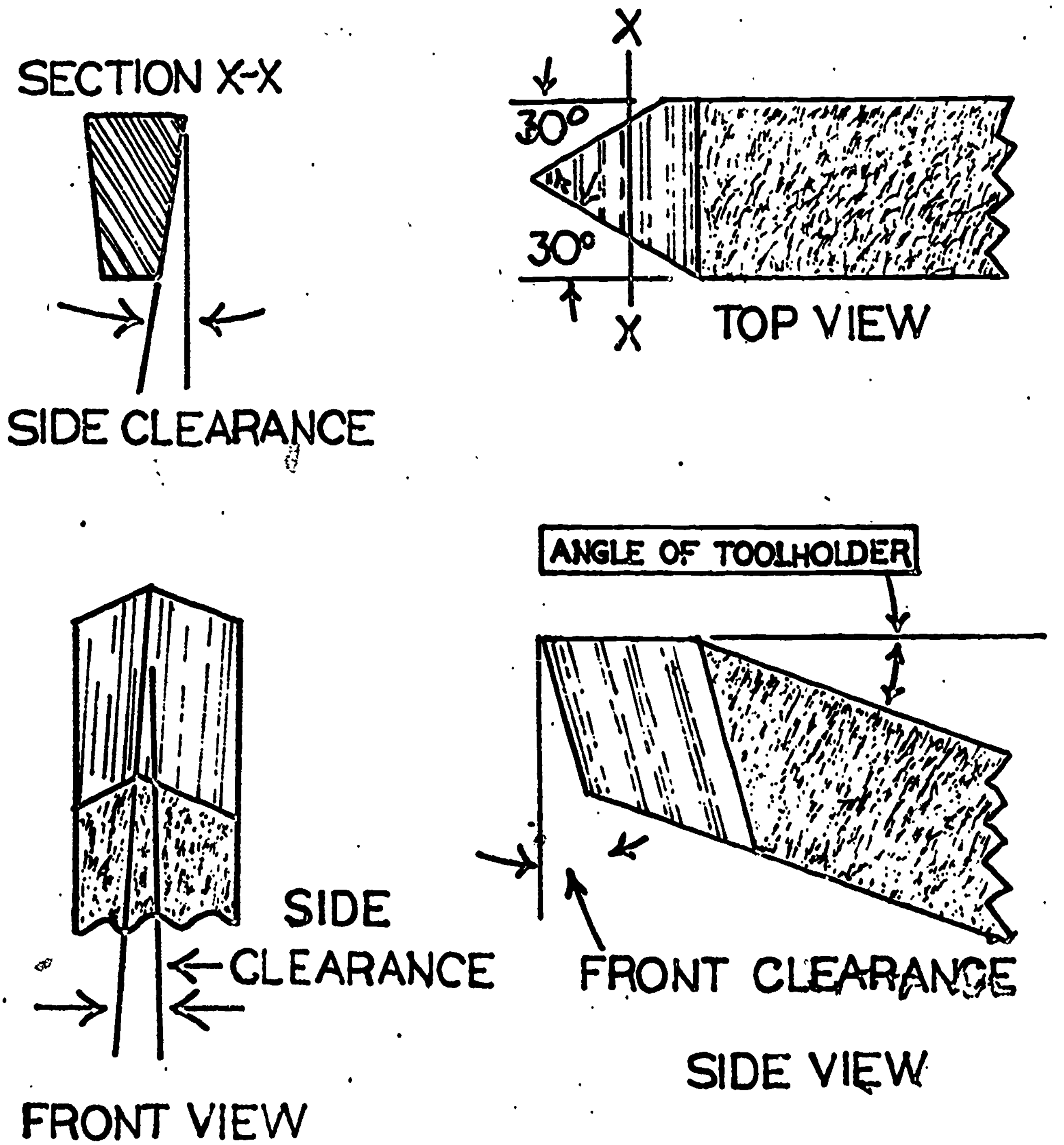
QUESTIONS:

1. What type of material is used for lathe and shaper tools?
2. What is the front clearance angle of a lathe tool for cutting steel?
3. What is the side clearance angle of a lathe tool for cutting steel?
4. What is the top rake angle of a lathe tool for cutting steel?
5. What is the side rake angle for a lathe tool for cutting steel?
6. How is a lathe tool mounted in relation to the work?
7. What are the advantages of a carbide - tipped tool?
8. Why should a tool be honed after it is ground?
9. How much front clearance should be on a boring tool?
10. How much front clearance should a shaper tool have?

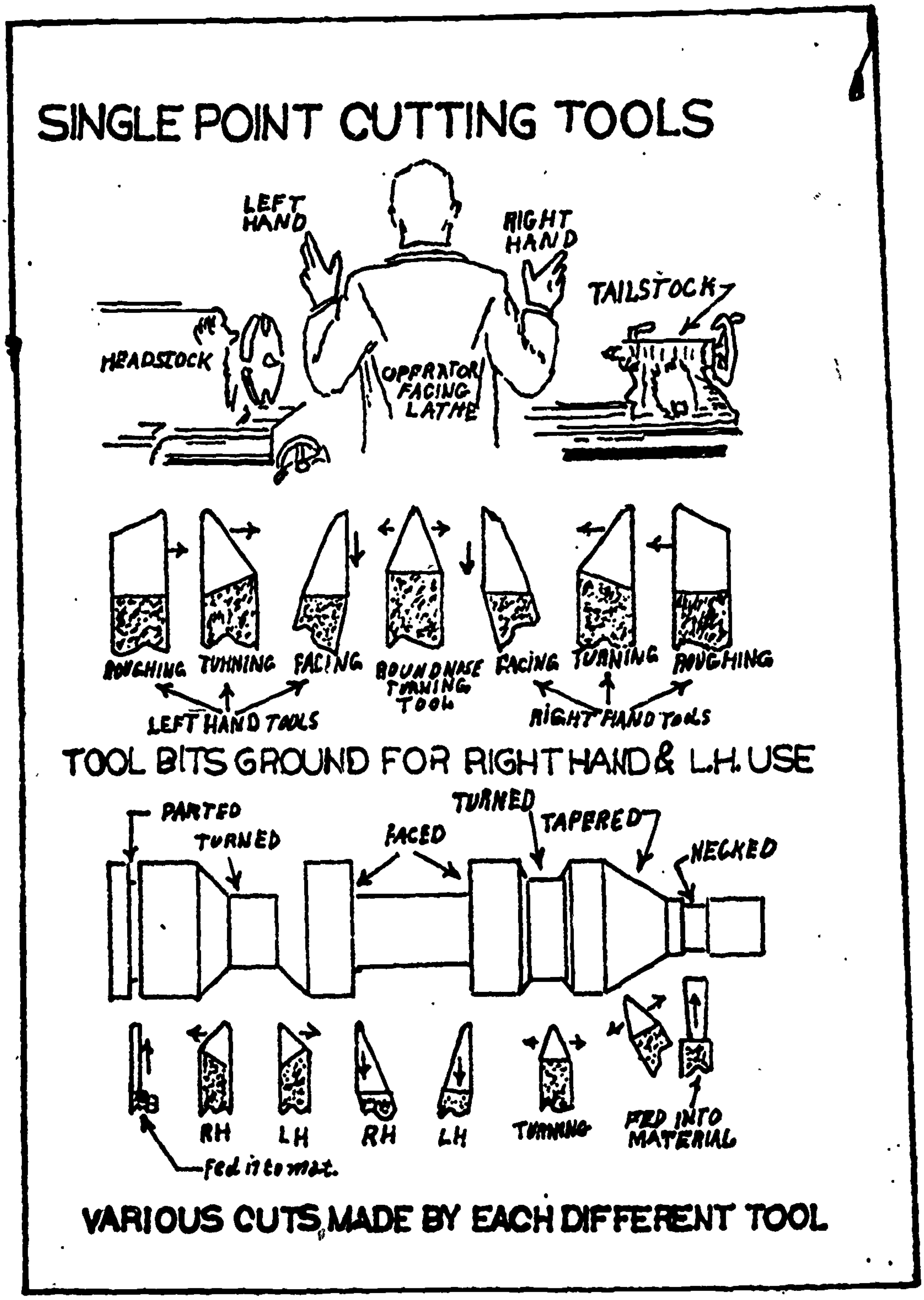
LATHE CUTTING TOOL



SINGLE PONT FORMNG TOOL

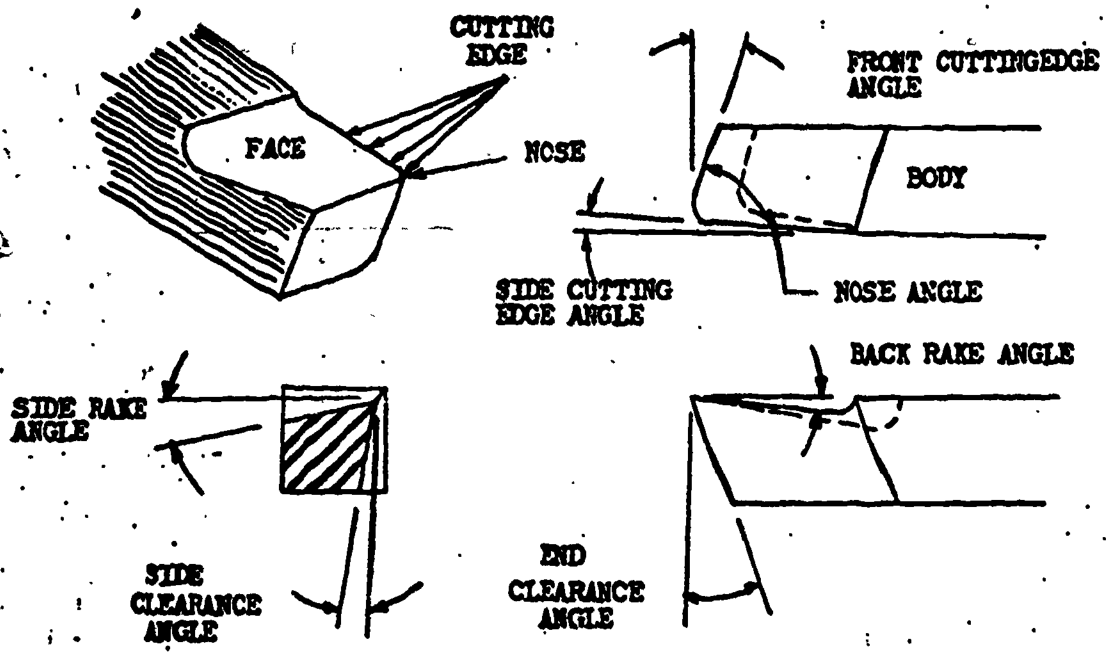


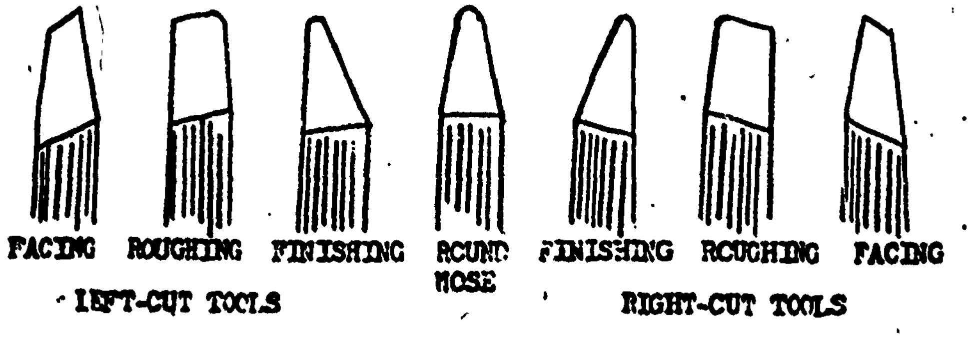
RIGHT-HAND 60° THREADNG TOOL


SINGLE POINT CUTTING TOOLS
LEFT HAND
RIGHT HAND
TAILSTOCK
HEADSTOCK
OPERATOR FACING LATHE
ROUGHING
TURNING
FACING
ROUNDNOSE TURNING TOOL
FACING
TURNING
ROUGHING
LEFT HAND TOOLS
RIGHT HAND TOOLS
TOOL BITS GROUND FOR RIGHT HAND & L.H. USE
PARTED
TURNED
FACED
TURNED
TAPERED
NECKED
RH
LH
RH
LH
TURNING
FED INTO MATERIAL
fed into mat.
VARIOUS CUTS MADE BY EACH DIFFERENT TOOL

M-5 18

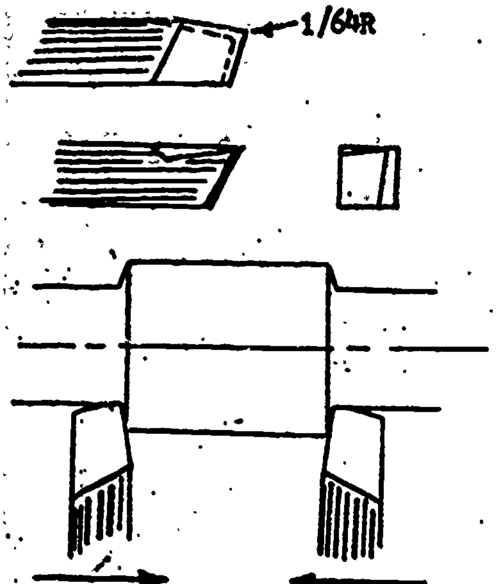
TOOL BIT NOMENCLATURE





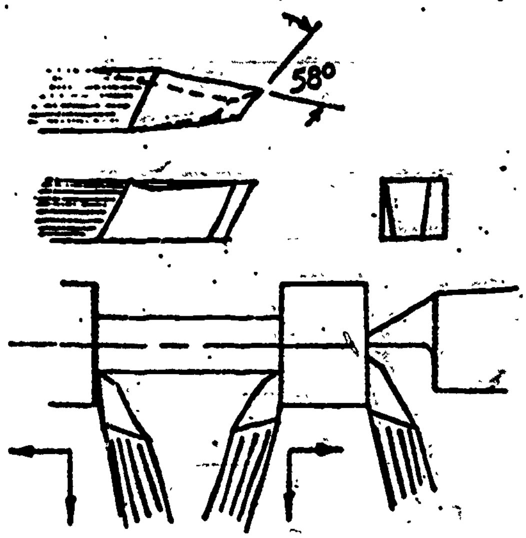
STANDARD CUTTING TOOL SHAPES

SINGLE POINT CUTTING TOOLS



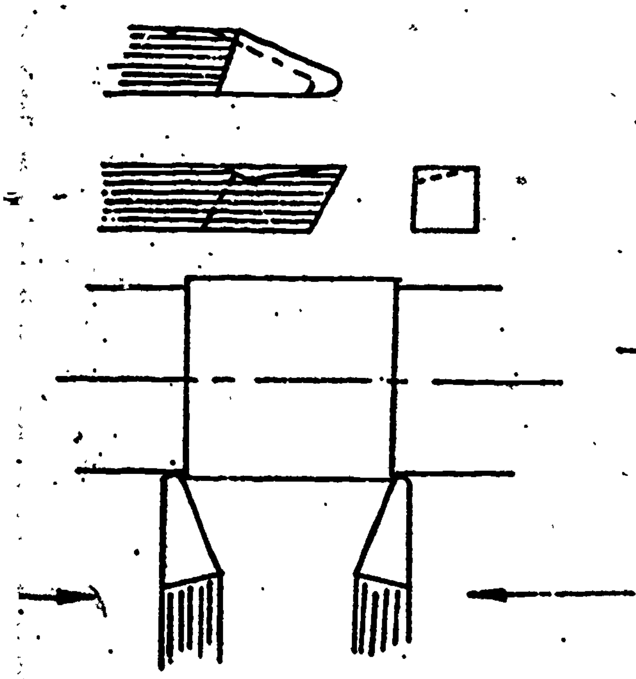
LEFT-CUT RIGHT-CUT

ROUGHING TOOLS



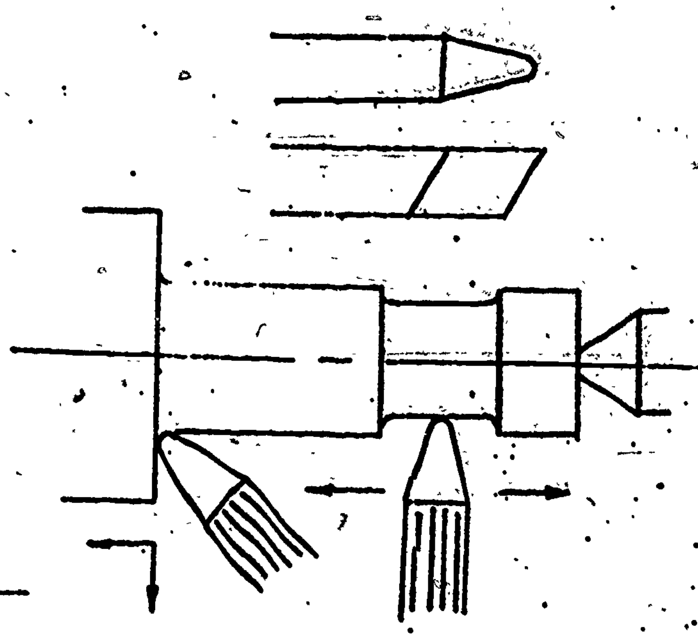
LEFT-CUT RIGHT-CUT

FACING TOOLS

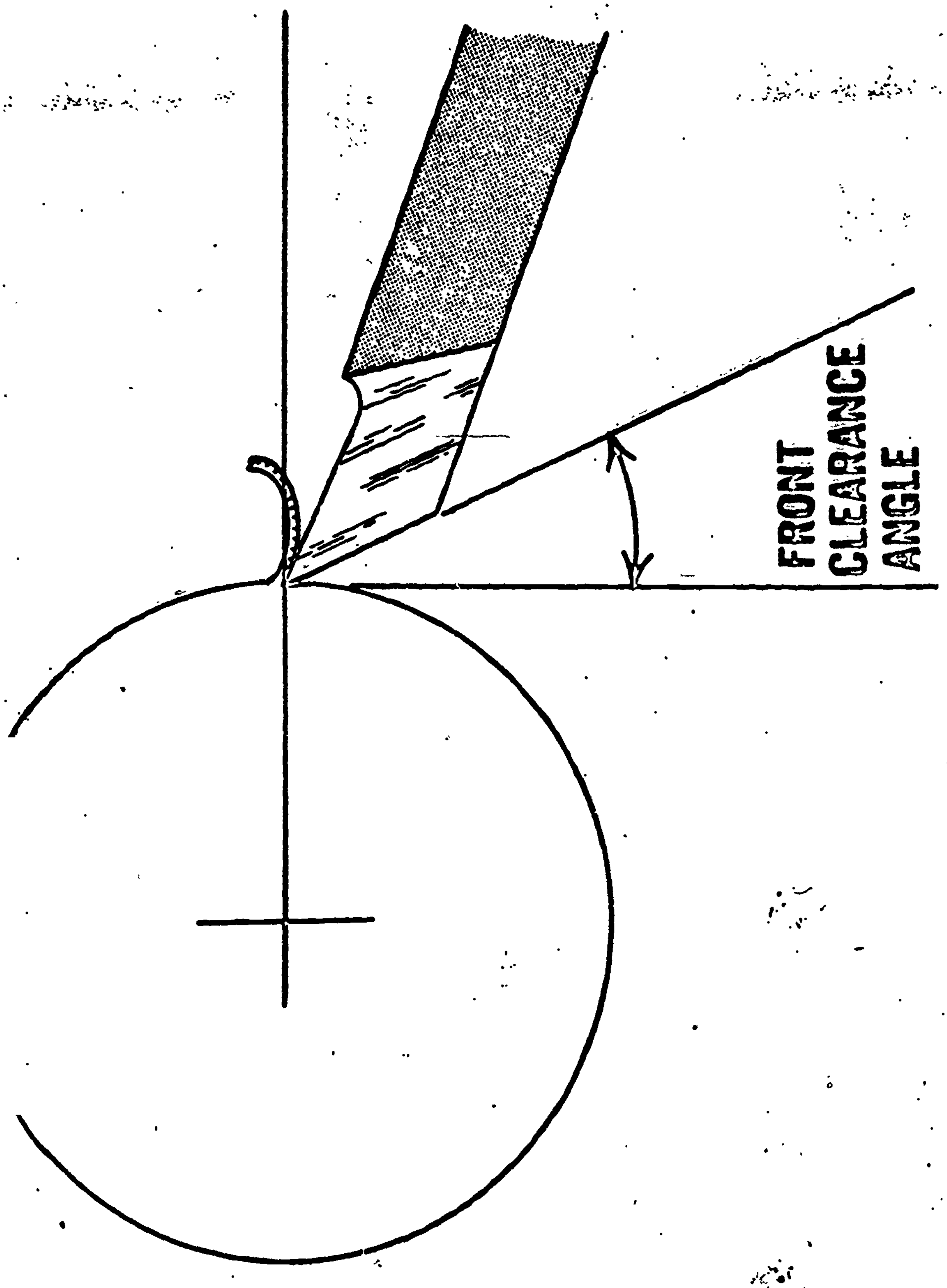


LEFT-CUT RIGHT-CUT

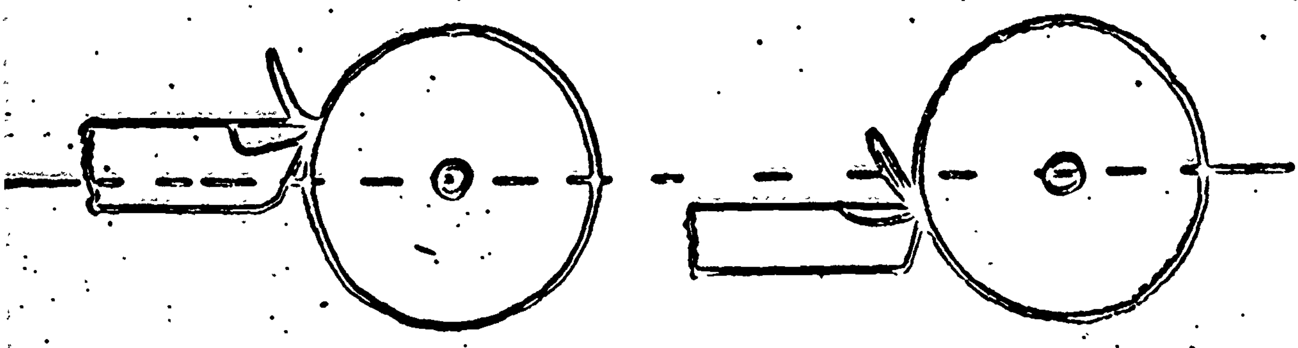
FINISHING TOOLS



ROUND NOSE TOOL


FRONT
CLEARANCE
ANGLE

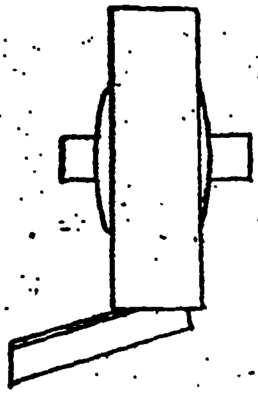
TOOLS SET WRONG



TOOL SET TOO HIGH WILL RUB

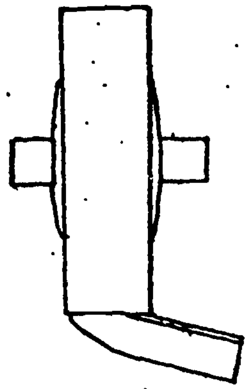
TOOL SET TOO LOW WILL BREAK

GRINDING A TOOL BIT



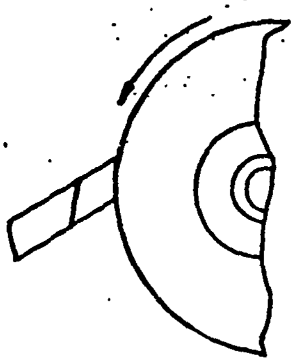
SIDE RELIEF AND
SIDE-CUTTING EDGE
ANGLE

STEP 1



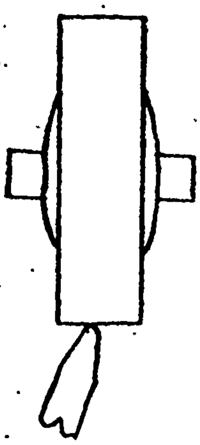
OPPOSITE
SIDE-RELIEF
ANGLE

STEP 2



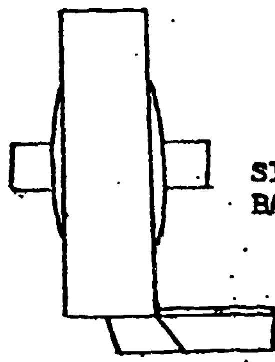
END-RELIEF ANGLE

STEP 3



ROUNDING OFF THE
NOSE

STEP 4



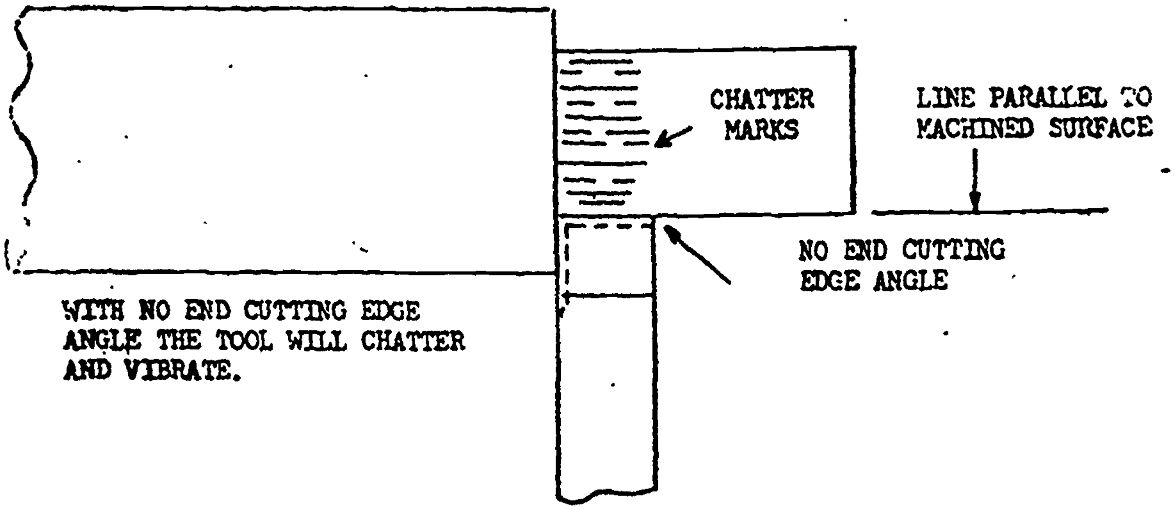
SIDE AND
BACK RAKE

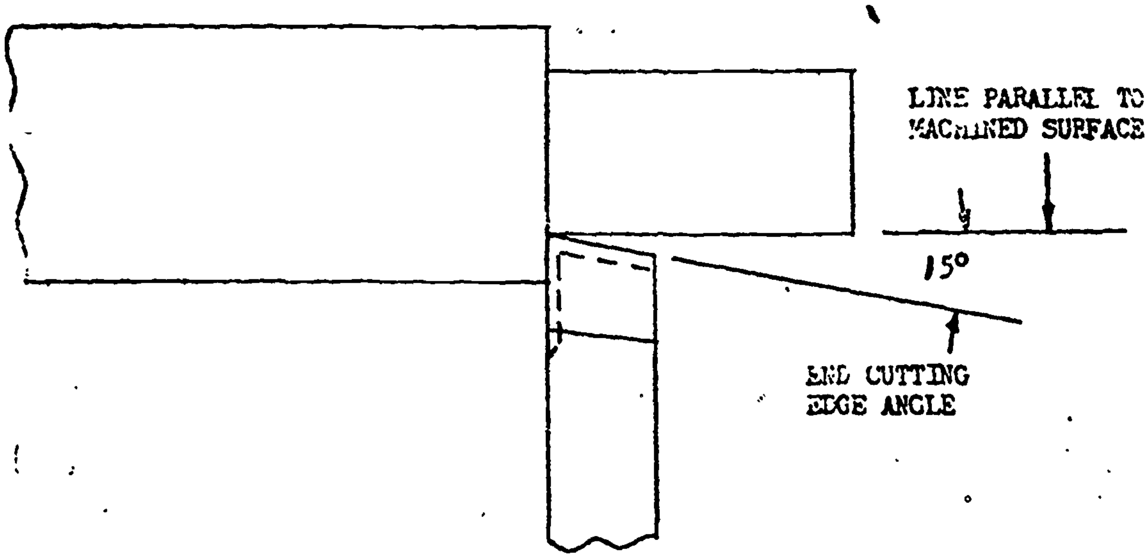
STEP 5

NOTE: RELIEF ANGLE BEING THE SAME
AS A CLEARANCE ANGLE

257

FUNCTION OF THE END CUTTING EDGE ANGLE





THE PRIME FUNCTION OF THE END CUTTING EDGE ANGLE IS TO PREVENT CHATTER AND VIBRATION.

TITLE: LATHES AND LATHE OPERATIONS

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To familiarize the student with lathes and lathe operations.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 6, pages 135-182.

DIRECTIONS: Read the above réference and answer the following questions.

QUESTIONS:

1. Explain the function of the following parts of a lathe:
   (A) Bed
   (B) Headstock
   (c) Tailstock
   (D) Carriage
   (E) Feed mechanism
   (F) Thread-cutting mechanism

2. How is feed expressed on a lathe?

3. How is the size of a lathe determined?

4. How is cutting speed expressed on a lathe?

5. Name the most commonly used lathe chucks and give the use of each.

6. What are collets and how are they used?

7. What attachments and accessories are used to turn work between centers?

8. What angle is cut on the end of lathe centers?

9. What is a mandrel?

10. What are the three methods of turning tapers?

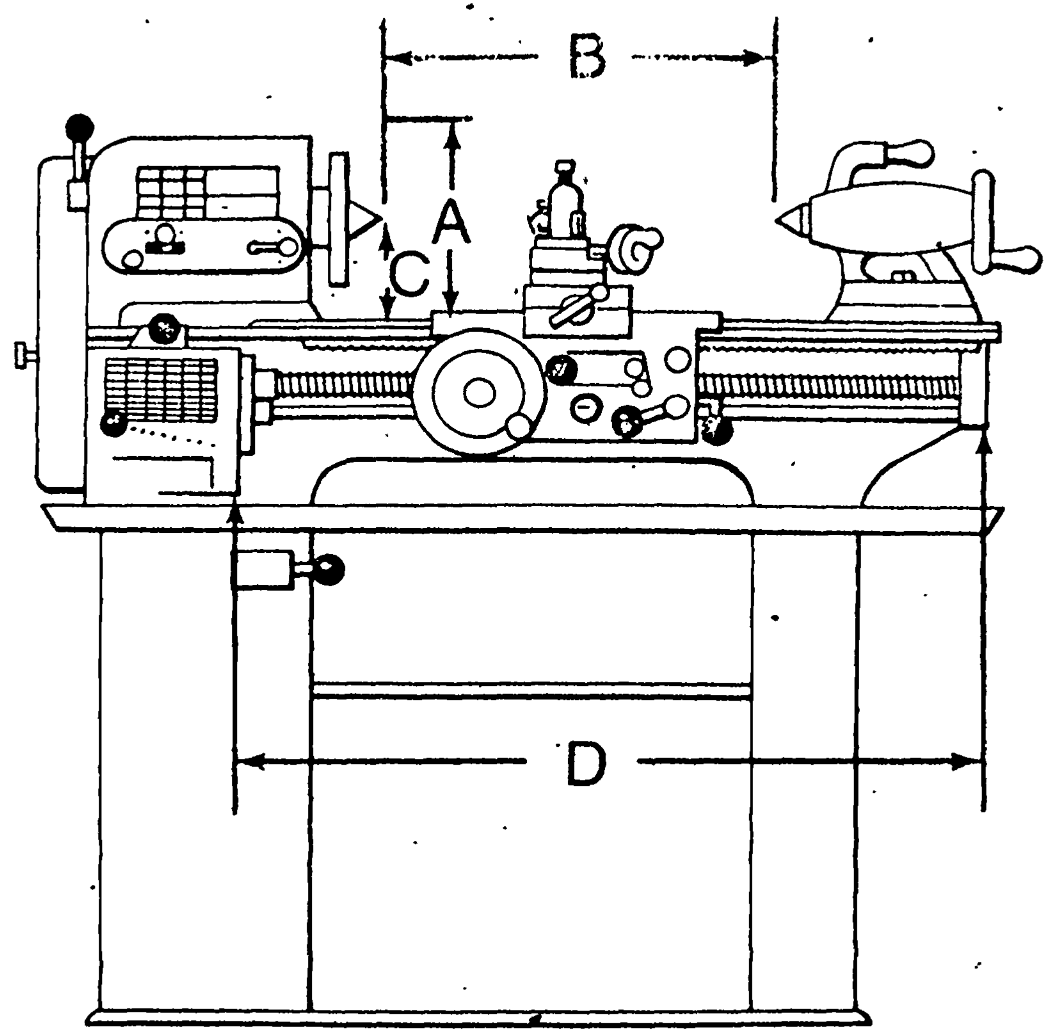
11. What are the advantages of using the taper attachment?

12. What is the operation of knurling?

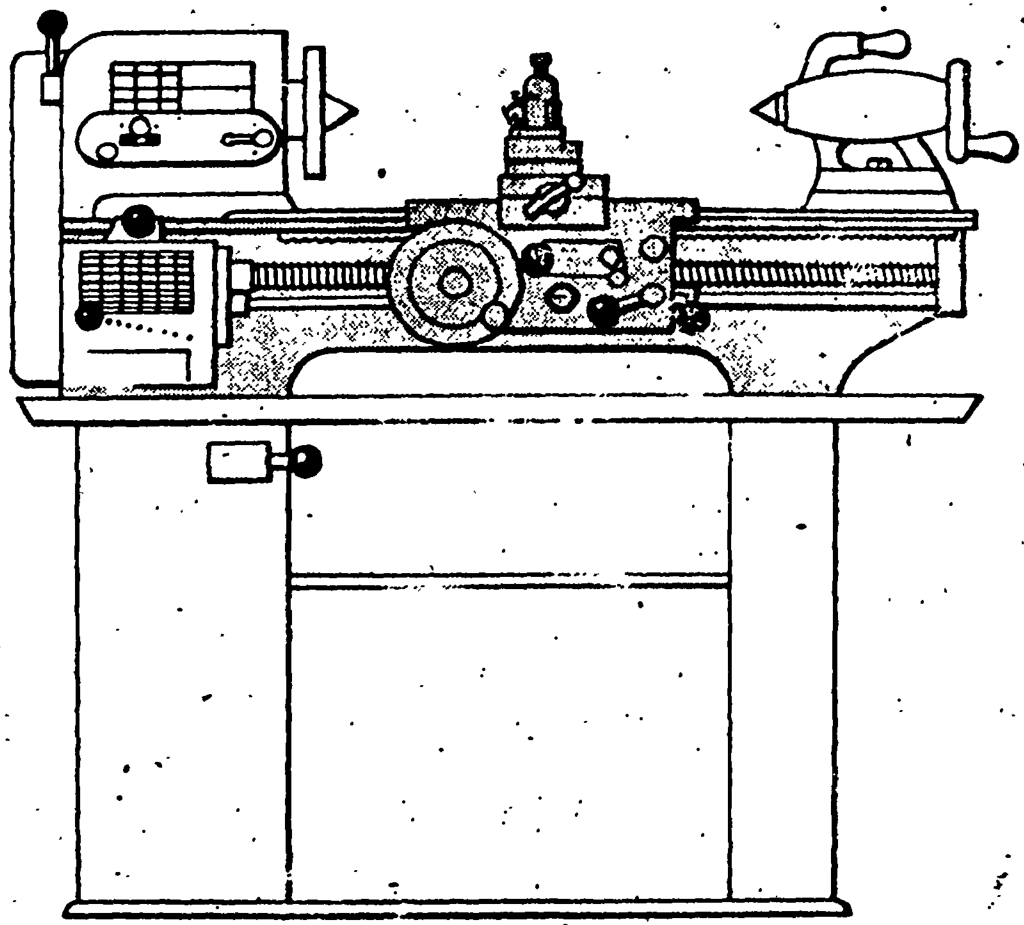
13. What is the procedure for machining a tapered shank?

14. List and explain the use of the most common lathe attachments.

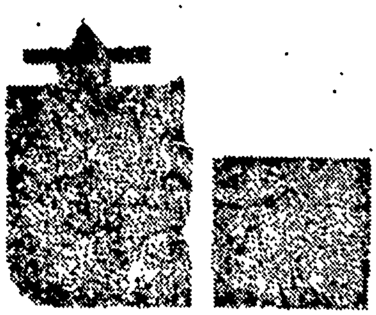
15. What is a good procedure for facing?

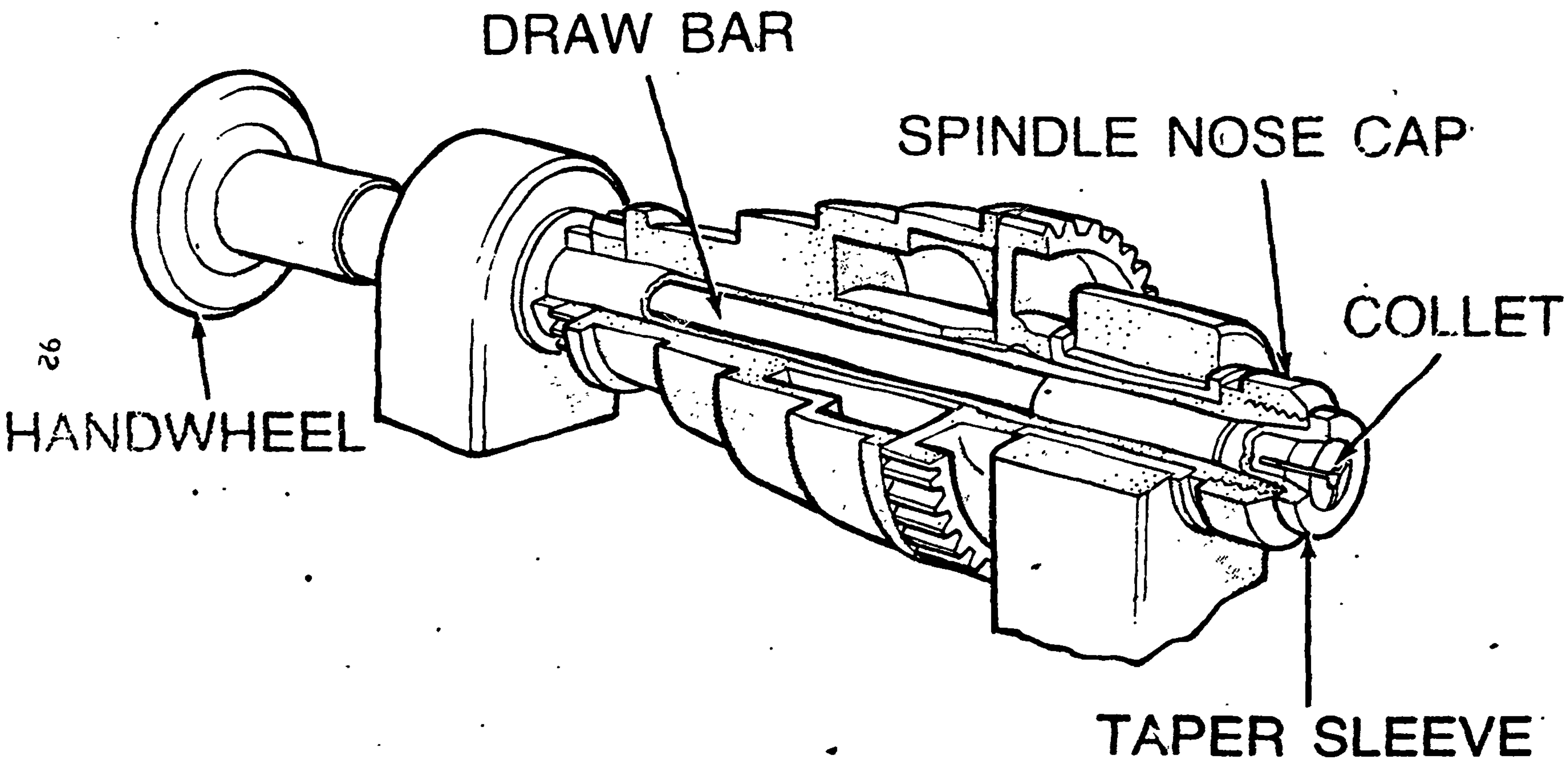


LATHE SIZES

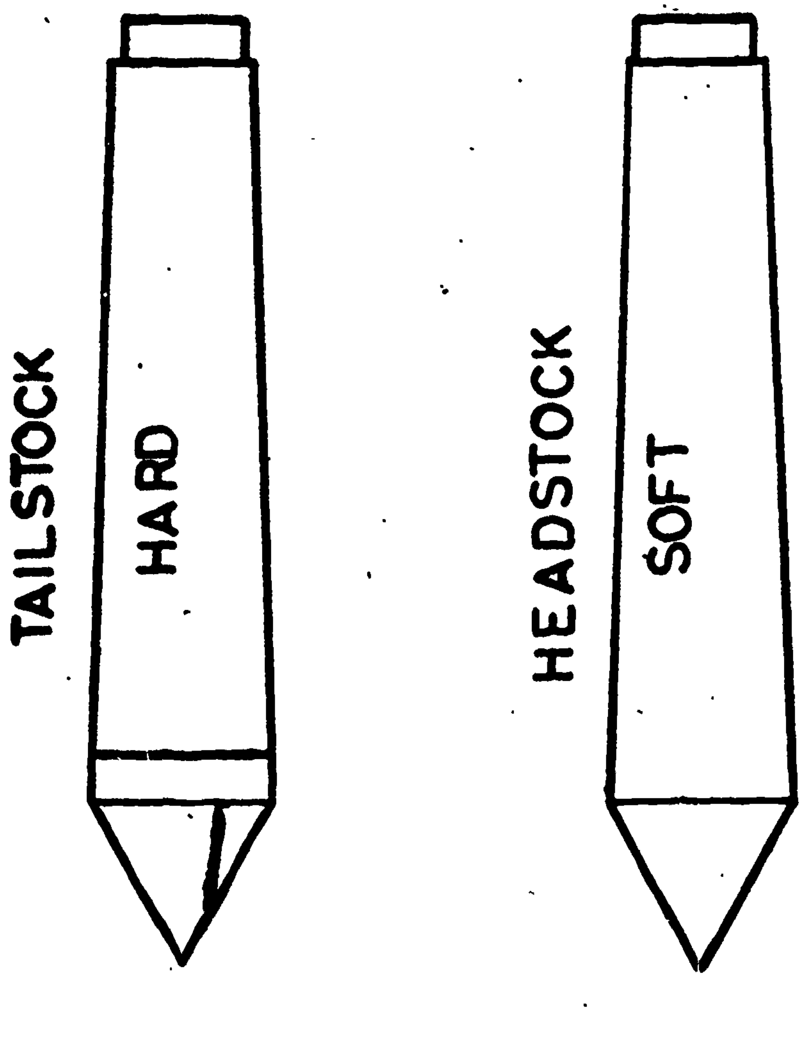


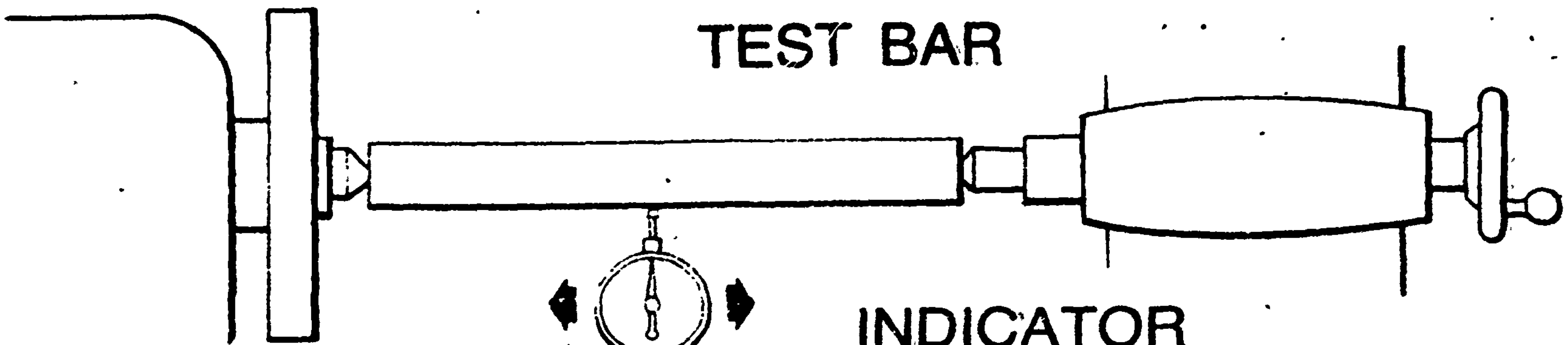
LATHE—MAJOR PARTS



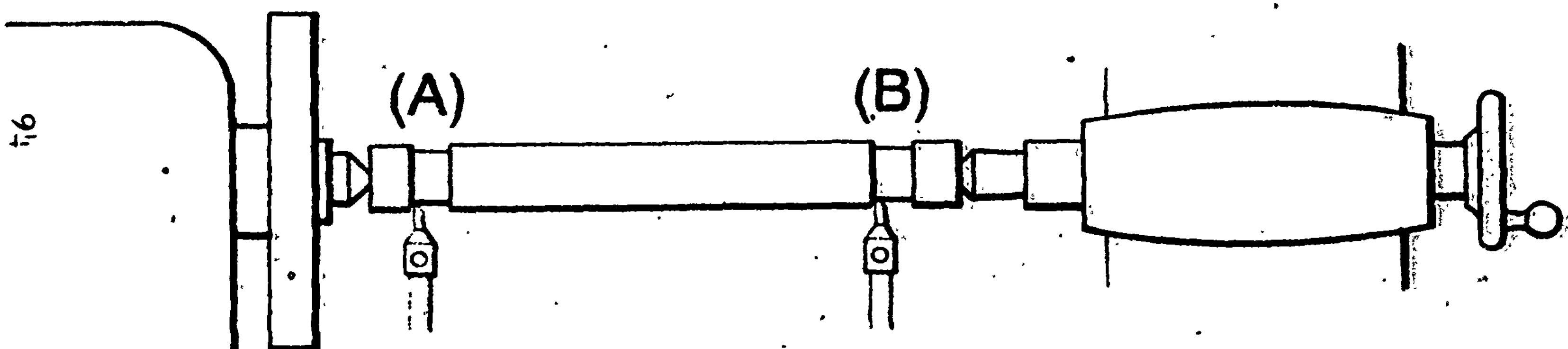


COLLET CHUCK ASSEMBLY


TAILSTOCK
HARD
HEADSTOCK
SOFT



ALIGNING LATHE CENTRE WITH TEST BAR AND INDICATOR

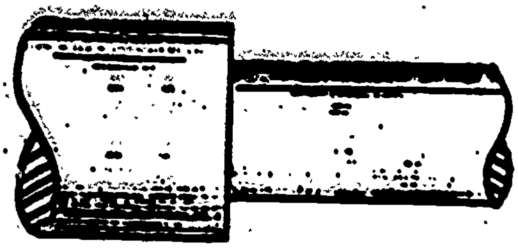


TESTING ALIGNMENT OF LATHE CENTRES BY 'CUT AND CHECK' METHOD

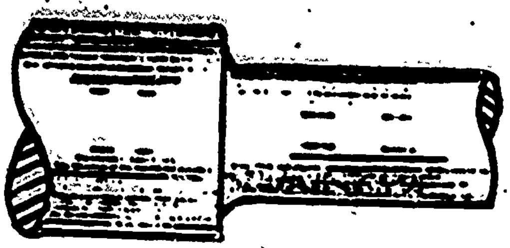
ALIGNMENT OF LATHE CENTRES

THREE TYPES OF SHOULDERS

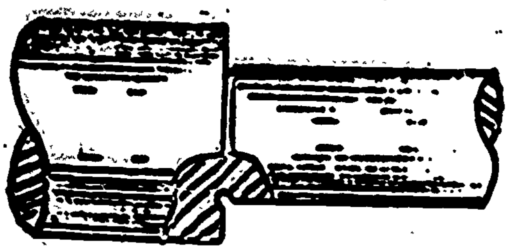
A



B



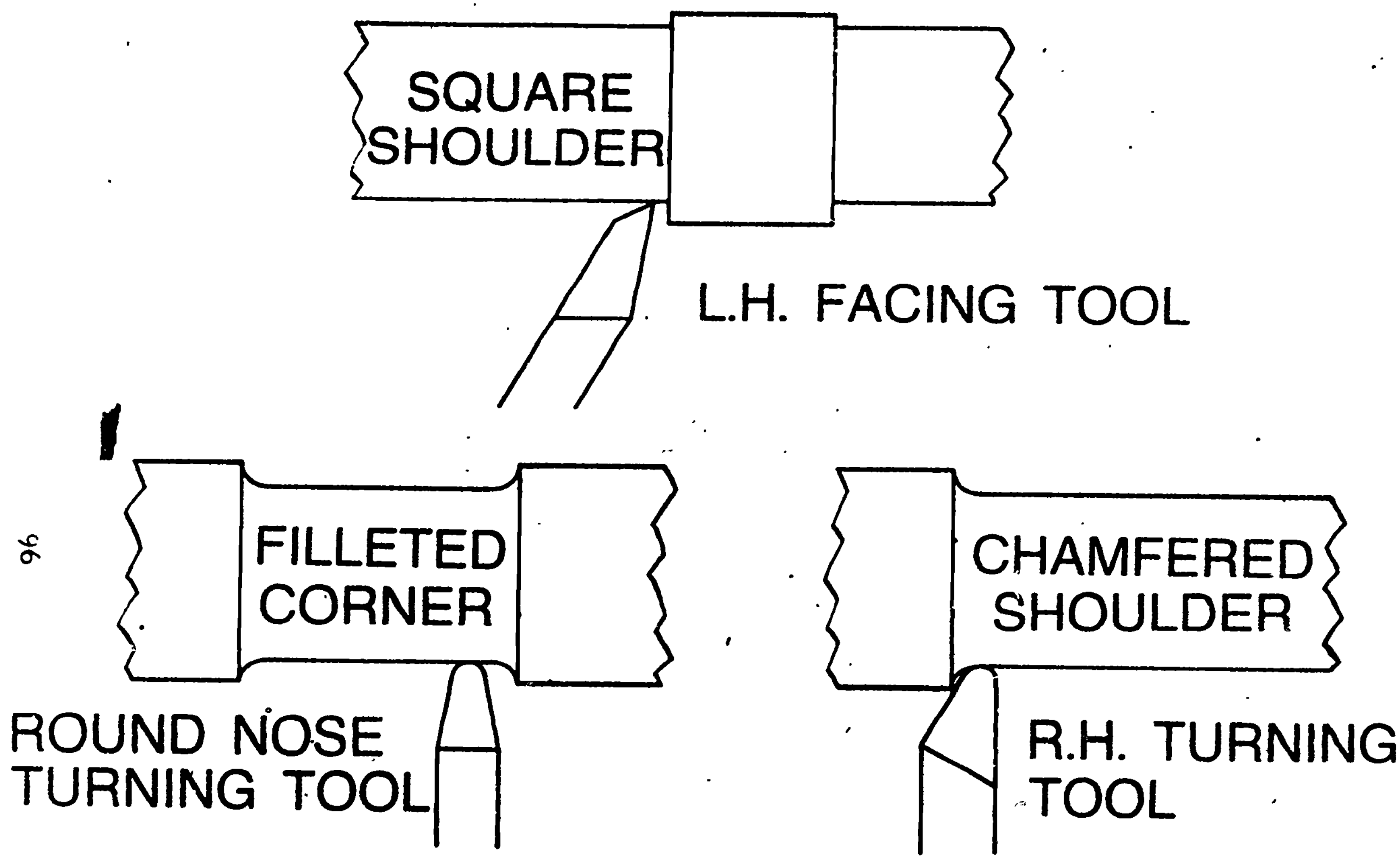
C



A. Square Shoulders

B. Radius Shoulder

C. Undercut Shoulder



TOOLBITS AND TURNING OPERATIONS
TYPES OF SHOULDERS

FORMULA TO FIND
CUTTING SPEEDS

$$\frac{D. \times R.P.M.}{4}$$

FORMULA TO FIND
R. P. M.

$$\frac{4 \times C.S.}{D.}$$

R.P.M. AND F.P.M.

BEST COPY AVAILABLE

FORMULA

$= \frac{12 \times F.P.M.}{\pi \times D}$ OR $RPM = \frac{4 \times F.P.M.}{DIA.}$

THUS

100 F.P.M. $\frac{4 \times 100}{1} = 400$ R.P.M.

100 F.P.M. $\frac{4 \times 100}{.5} = 800$ R.P.M.

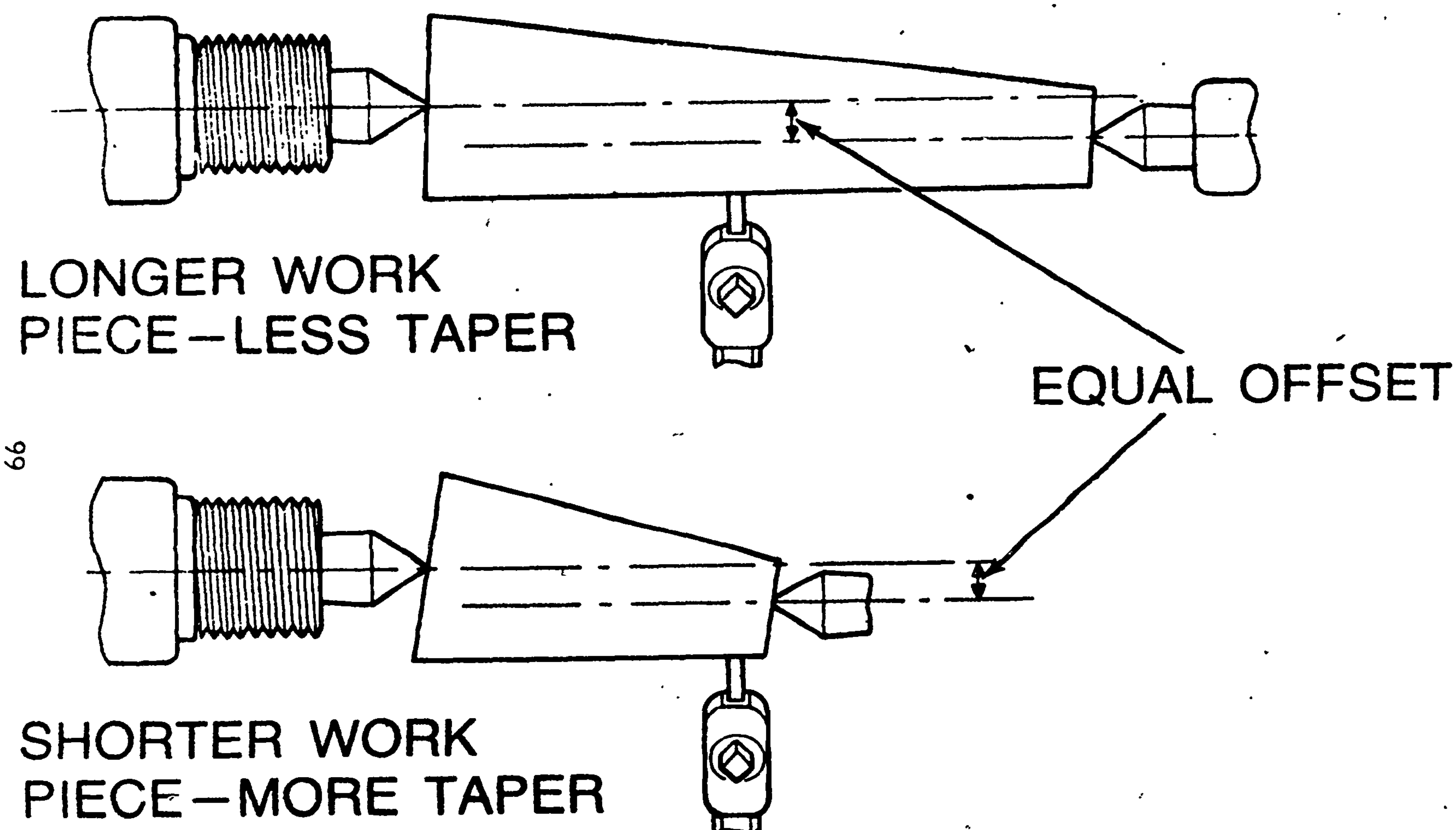
100 F.P.M. $\frac{4 \times 100}{.25} = 1600$ R.P.M.

M. $= \frac{D \times R.P.M.}{4}$ $\frac{1'' \times 400}{4} = 100$ F.P.M.

400 R.P.M. $\frac{1'' \times 400}{4} = 100$ F.P.M.

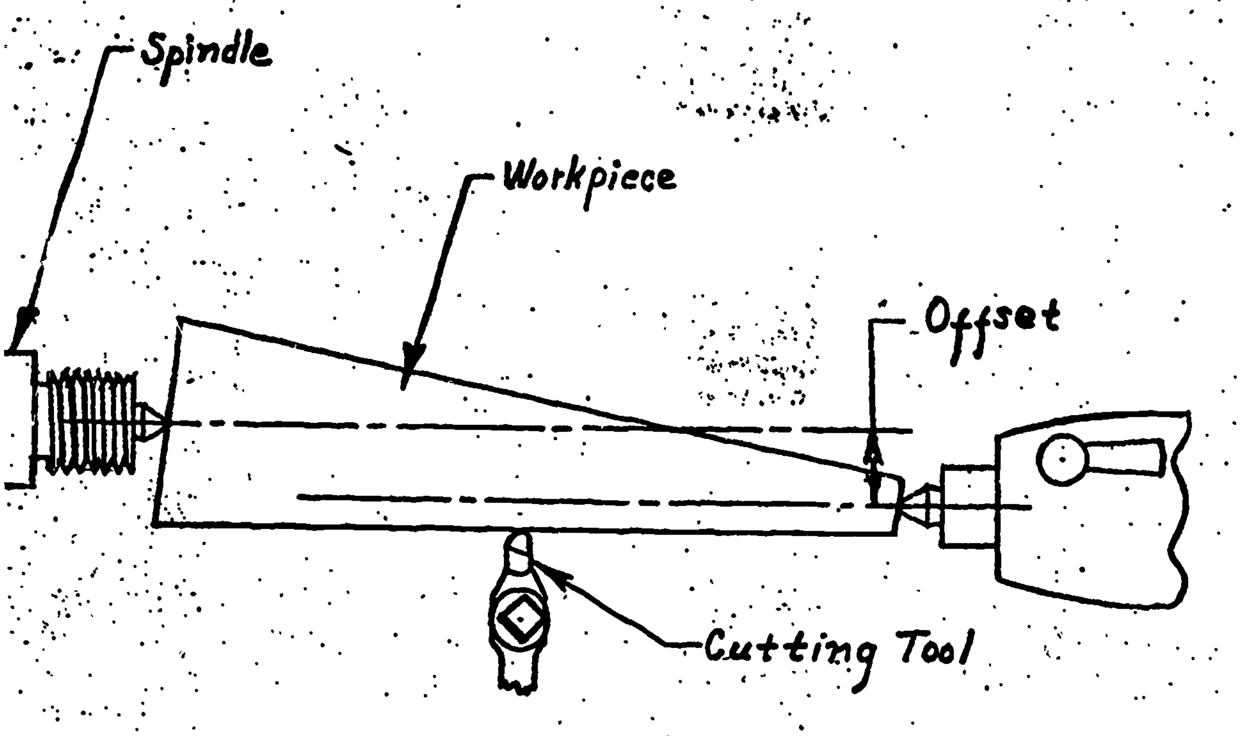
800 R.P.M. $\frac{.5 \times 800}{4} = 100$ F.P.M.

1600 R.P.M. $\frac{.25 \times 1600}{4} = 100$ F.P.M.



TAPER VARIES WITH LENGTH

ENGINE LATHE



Producing A Taper By Offsetting The Tailstock

Only Method Where Cutting Tool Moves Parallel To The Bed Ways

FORMULA TO FIND TAILSTOCK OFFSET FOR TURNING TAPER

$$\text{OFFSET} = \frac{L. \times T.P.F.}{12 \times 2}$$

$$\frac{12 \times 5}{12 \times 2} = \frac{6.0}{24}$$

$$24\overline{)6.0} = .25$$

48
120
120

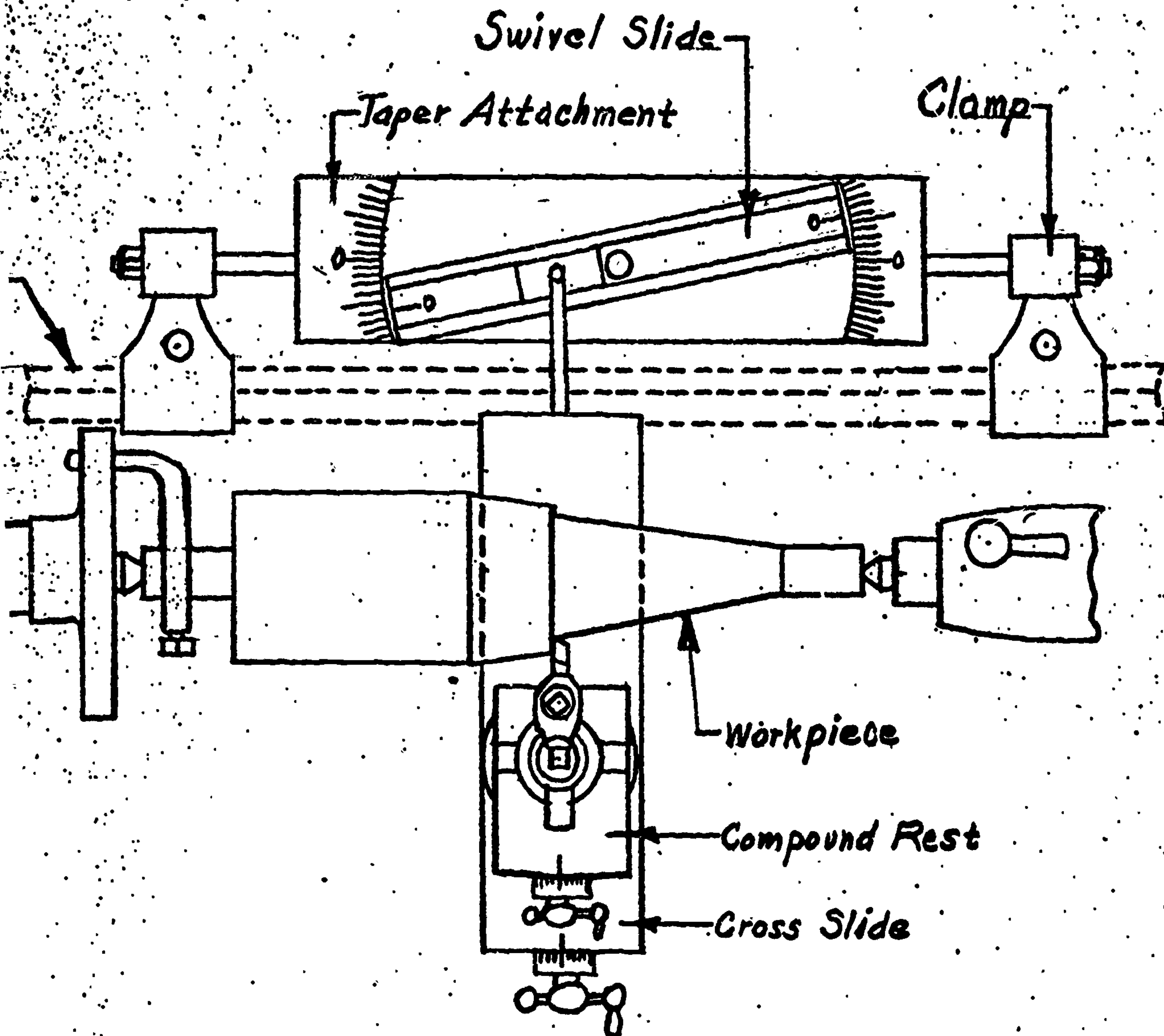
OR $\frac{1}{4}$

OR

$$\text{OFFSET} = \frac{L. \times T.P.I.}{2}$$

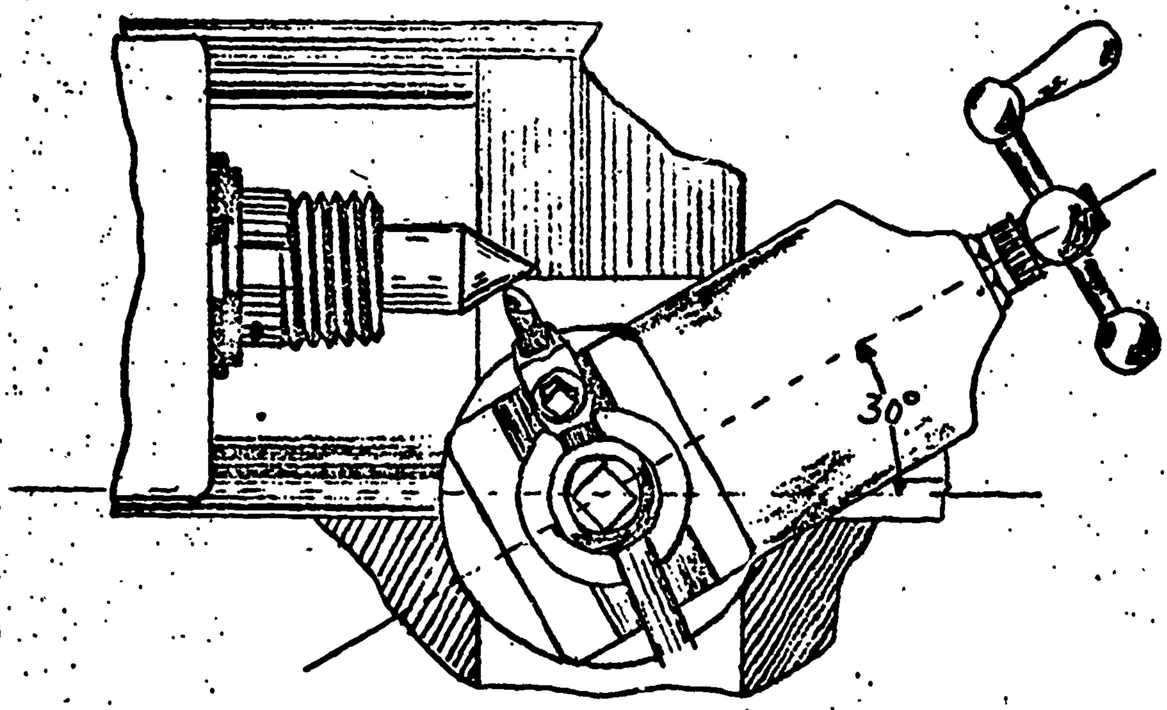
TO FIND TAPER PER INCH DIVIDE TAPER PER FOOT BY 12

ENGINE LATHE



rning A Taper Using A Taper Attachment

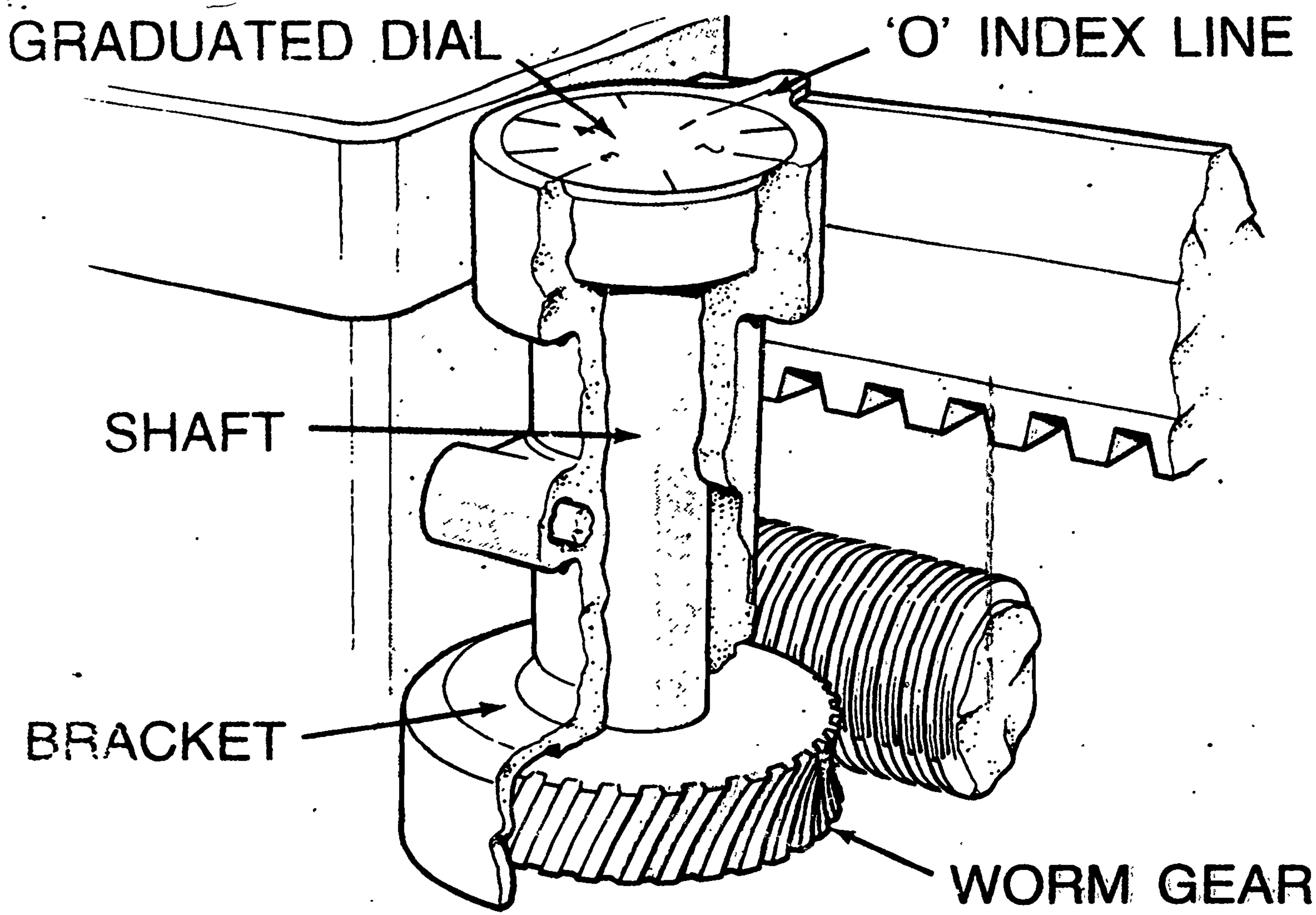
USING THE COMPOUND FOR TURNING A TAPER



LATHE CENTER 60° INCLUDED ANGLE

NOTE: POSITION OF COMPOUND IN RELATION TO CENTER LINE OF MACHINE.

76



THREAD DIAL INDICATOR

TITLE: TO FIND THE RPM AND CUTTING SPEED OF A LATHE

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To give the student practice in solving problems used in calculating the cutting speed or the R.P.M.'s of a lathe.

REFERENCE: (1) Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
(2) Axelrod Aaron, Machine Shop Mathematics: New York: McGraw - Hill Book Co., Inc.

DIRECTIONS: Read the above references, study the following examples and work the problems below.

Cutting speed is the rate expressed in feet per minute (FPM) at which a point on the circumference of the work passes the tool bit. To determine the cutting speed (CS), in surface feet per minute (FPM), all that is needed is the diameter (D) and the revolutions per minute (RPM) of the work piece.

Revolutions per minute must be determined to get the correct cutting speed for the material being machined. The revolutions per minute (RPM) may be determined when the cutting speed (CS) and the diameter (D) are known.

Examples: Find the cutting speed of a 3/4" diameter piece of steel revolving at 240 revolutions per minute.

Solution: $CS=\frac{DXRPM}{4}=\frac{3/4x240}{4}=45FPM$

Find the revolutions per minute of a 2" diameter steel shaft when the cutting speed is 80' per minute

Solution: $RPM=\frac{4xCS}{D}=\frac{4x80}{2}=160$ RPM

PROBLEMS:

1. A piece 2½" in diameter is being turned in a lathe. Find the cutting speed if the work is revolving at 560 revolutions per minute.

2. A piece of 1" cold-rolled steel is being turned at 158 revolutions per minute. Find the cutting speed.

3. How many revolutions per minute should a piece of steel 3" in diameter make in order that the lathe tool will cut at the rate of 60' per minute?

4. How fast should a piece of $2\frac{1}{2}$" diameter brass rotate in order that the tool may have a cutting speed of 140' per minute?

5. A piece of tool steel that measures $1\frac{1}{4}$" in diameter is to be made into a punch. How fast should the work rotate in order that the tool will cut at the rate of 35' per minute?

6. A safe cutting speed for lathe tools made of carbon steel is 30' per minute. Find the revolutions per minute required in order that the lathe tool may have a cutting speed of 30' per minute while cutting a piece of stock $4\frac{1}{2}$" in diameter of mild steel.

7. A lathe hand is required to make a number of steel taper pins out of 5/8" diameter stock. How fast should the work revolve in order that the cutting speed may be 55' per minute?

8. A piece rate worker has a job of boring brass bushings whose inside diameter is 1 5/8". How fast should he run the lathe so that the cutting speed of his tool will be 95' per minute.

9. A piece of 7/8" diameter steel stock is being revolved at 120 revolutions per minute. Find the cutting speed.

10. A forging for a crankpin measures $6\frac{1}{4}$' in diameter in the rough is to be machined. The cutting speed should not be more than 45' per minute in order that the tool may "stand up" a reasonable length of time. How fast can the machinist drive his lathe on this job.

TITLE: MACHINE TAPERS

UNITE: TAPERS

OCCUPATION: MACHINIST

OBJECTIVE: To acquaint the student with the standard types of machine tapers.

REFERENCE: Anderson - Tatro. Shop Theory. New York: McGraw-Hill Book Co. , Inc. Chapter 8, pages 200-209.

DIRECTIONS: Read the above reference and answer the following questions:

QUESTIONS:

1. What is meant by a self holding taper?
2. What type taper is used on taper shanked drills?
3. What is the taper per foot of standard taper pins?
4. Where is the Brown and Sharpe taper used most often?
5. What type of taper is used on most new model milling machines?
6. What is the taper per foot of the Brown and Sharpe taper?
7. What is the taper per foot of the Jarno taper?
8. How is the size of Morse tapers denoted?
9. How is the fit of a taper checked?
10. What is the purpose of the tang on a taper shanked drill?

ASSIGNMENT SHEET

TITLE: TAPER PROBLEMS

UNIT: TAPERS

OCCUPATION: MACHINIST

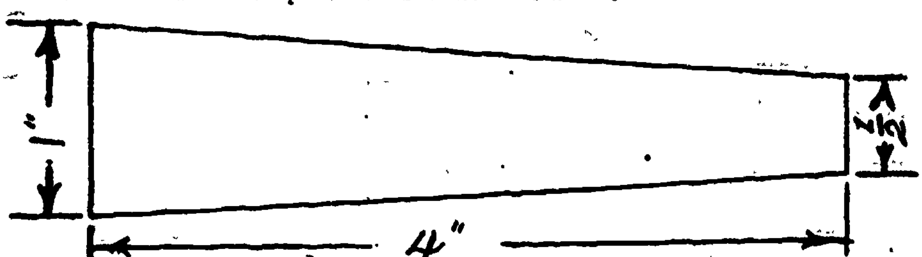
OBJECTIVE: To give the student practice in solving taper problems.

REFERENCE: 1. Axelrod, Aaron. Machine Shop Mathematics. New York. McGraw-Hill Book Co., Inc.

2. INFORMATION SHEET-Taper Formulas.

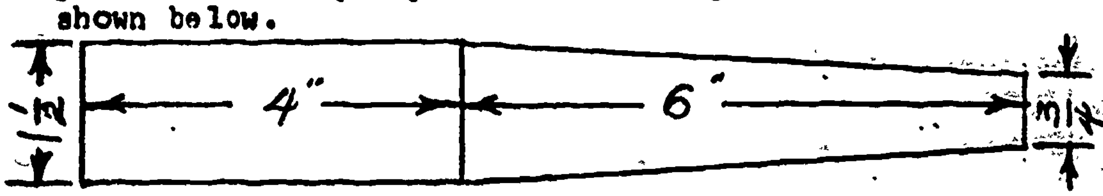
DIRECTIONS: Read the above references and using the information sheet as a guide work the following problems.

PROBLEMS: 1. Find the taper per inch and the taper per foot of the piece shown below.

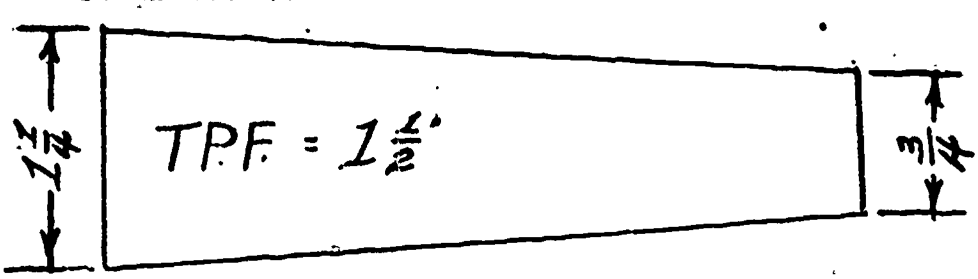


2. A tapered reamer is .625" in diameter at the large end and .600" at the small end. The over all length of the reamer is 7½" and the part that is tapered is 5" long. Find the taper per inch and the taper per foot.

3. Find the taper per foot of the piece shown below.



4. Find the lengh of taper of the piece shown below.



ASSIGNMENT SHEET

TITLE: TYPES OF SCREW THREADS

UNIT: SCREW THREADS

OCCUPATION: MACHINIST

OBJECTIVE: To familiarize the student with the different types of screw threads, their uses and how they are made.

DIRECTIONS: Read the above reference, and answer the following questions.

QUESTIONS:

1. What are the common uses of screw threads?
2. How are thread dimensions specified on a blueprint?
3. How are threads c hecked for accuracy?
4. What are the three types of taps in a set?
5. What is a die stock?
6. What is a tap wrench?
7. How do pipe threads differ from other type threads?
8. What is the difference in the two forms of the American National screw thread form?
9. How are most commerical threads produced?
10. What class thread would you find in a machine shop?

TERMS DEFINE:

1. Screw thread
2. Pitch Dia
3. Pitch
4. Lead
5. Crest

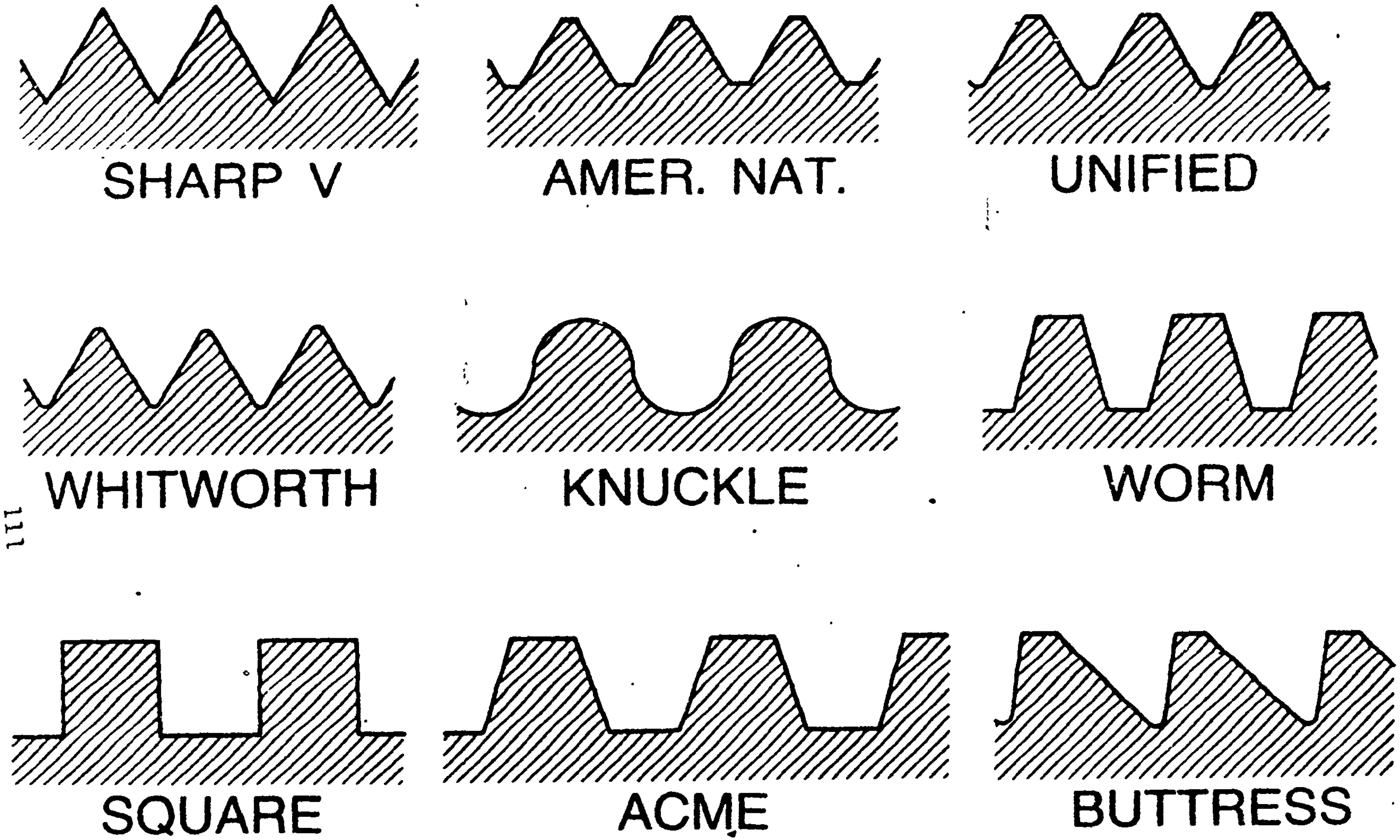
6. Thread angle

7. Left hand thread

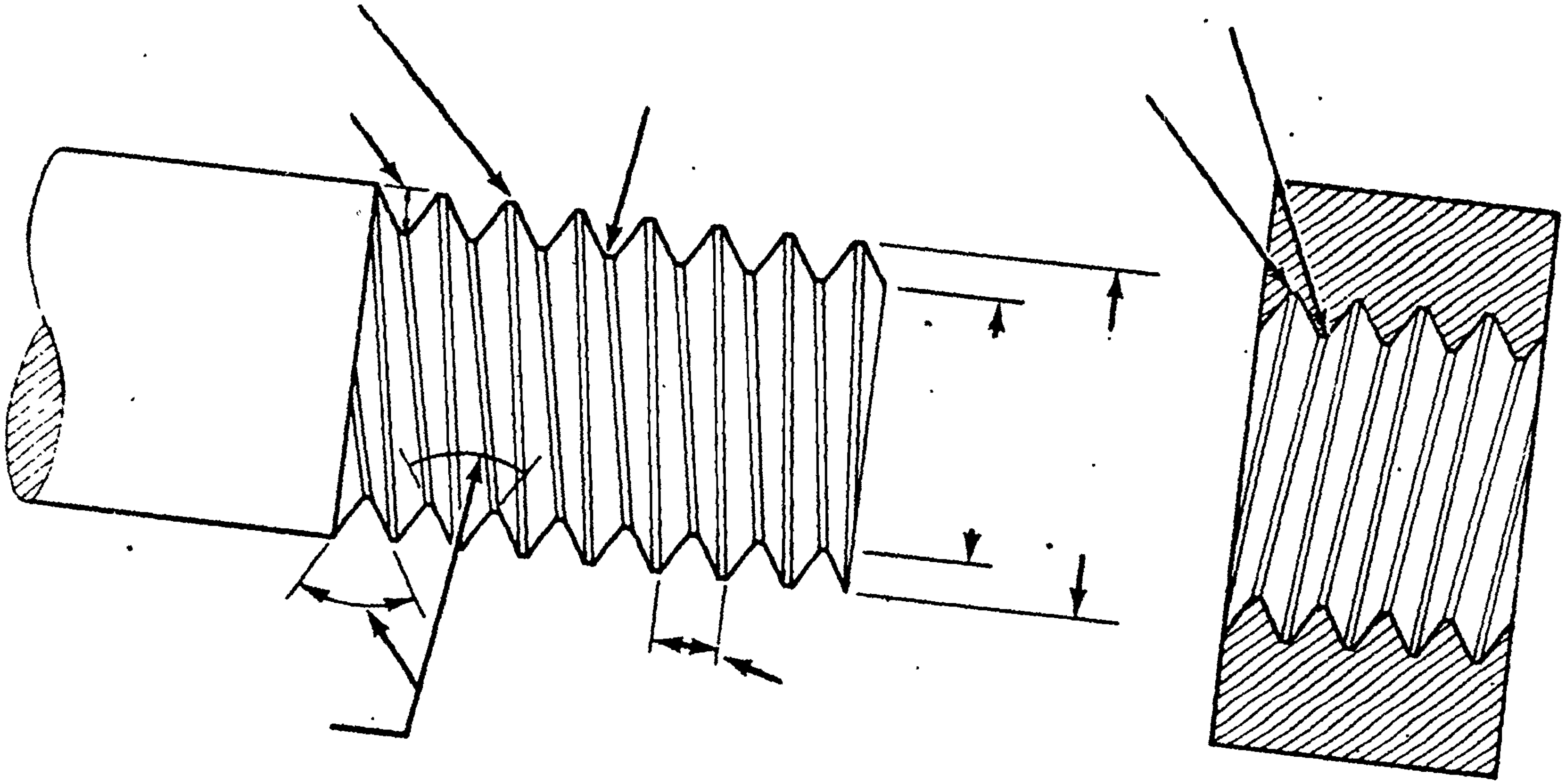
8. Fit

9. Tolerance

10. Allowance



SCREW THREAD FORMS



THREAD NOMENCLATURE

CREST

ROOT

CREST

ROOT

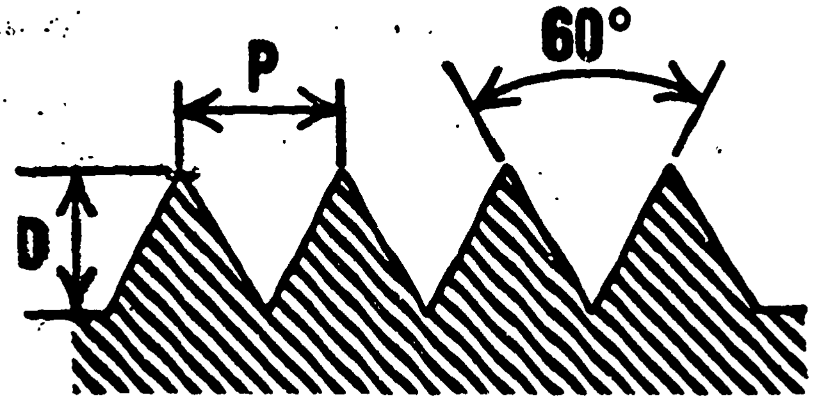
DEPTH

MINOR DIA.

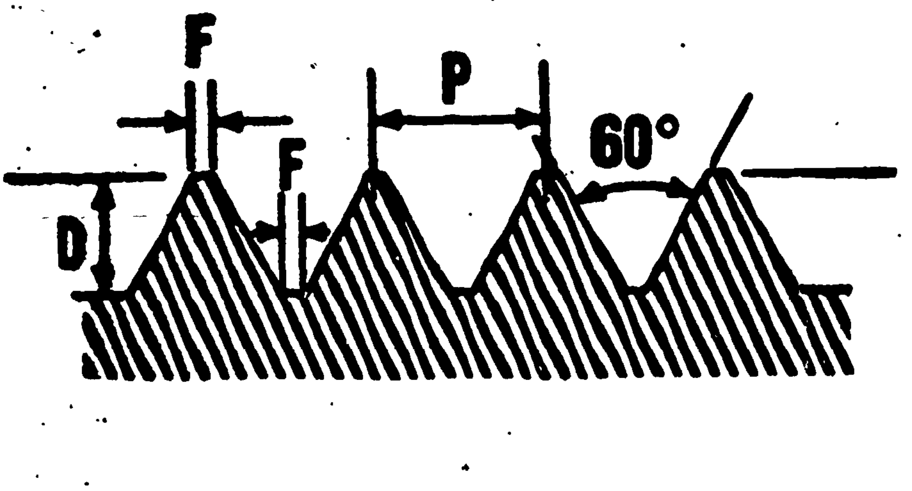
MAJOR DIA.

PITCH

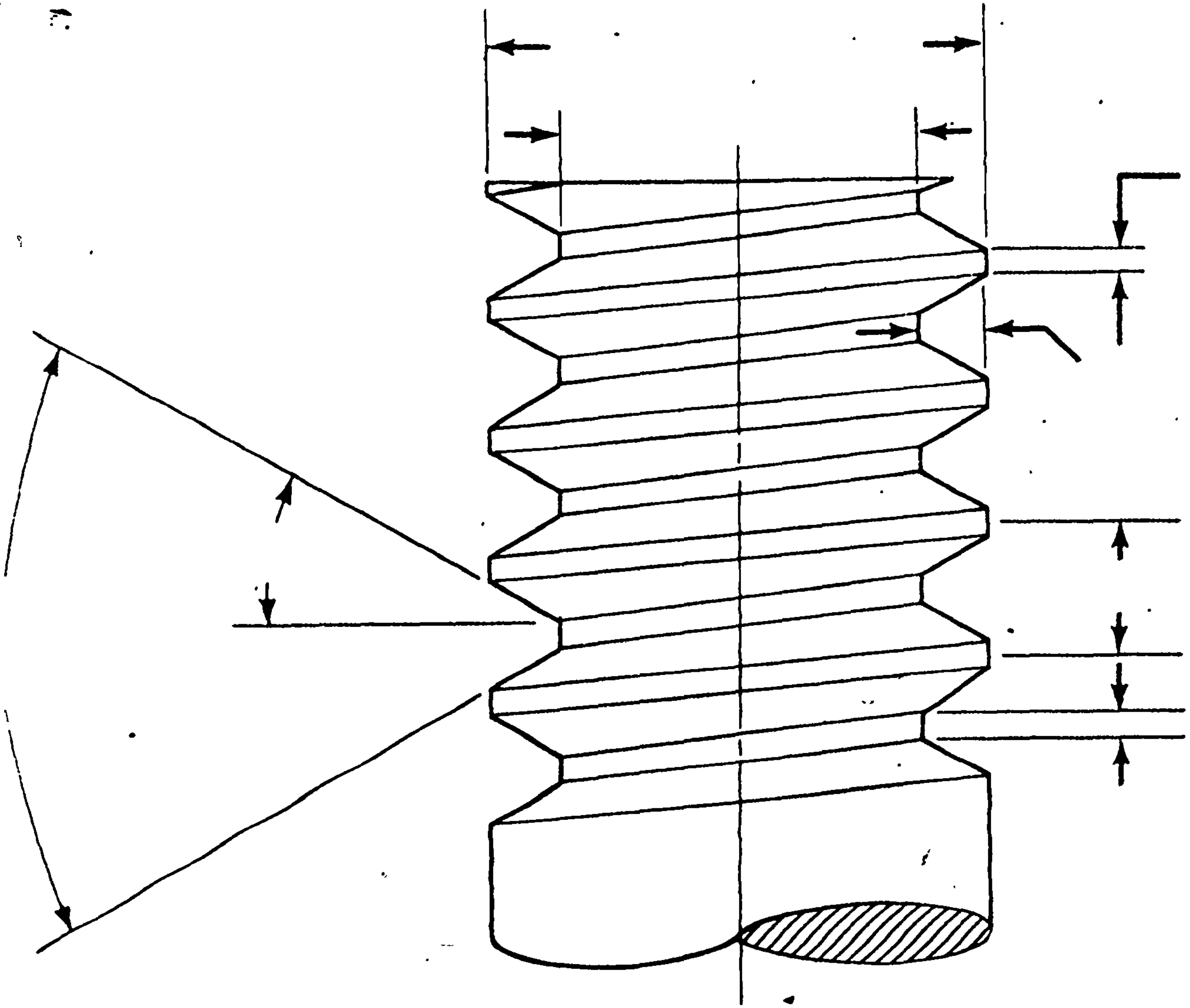
THREAD ANGLE



60° "V" THREAD



AMERICAN NATIONAL OR NATIONAL FORM THREAD



THREAD TERMINOLOGY

MAJOR DIA.

MINOR DIA.

CREST

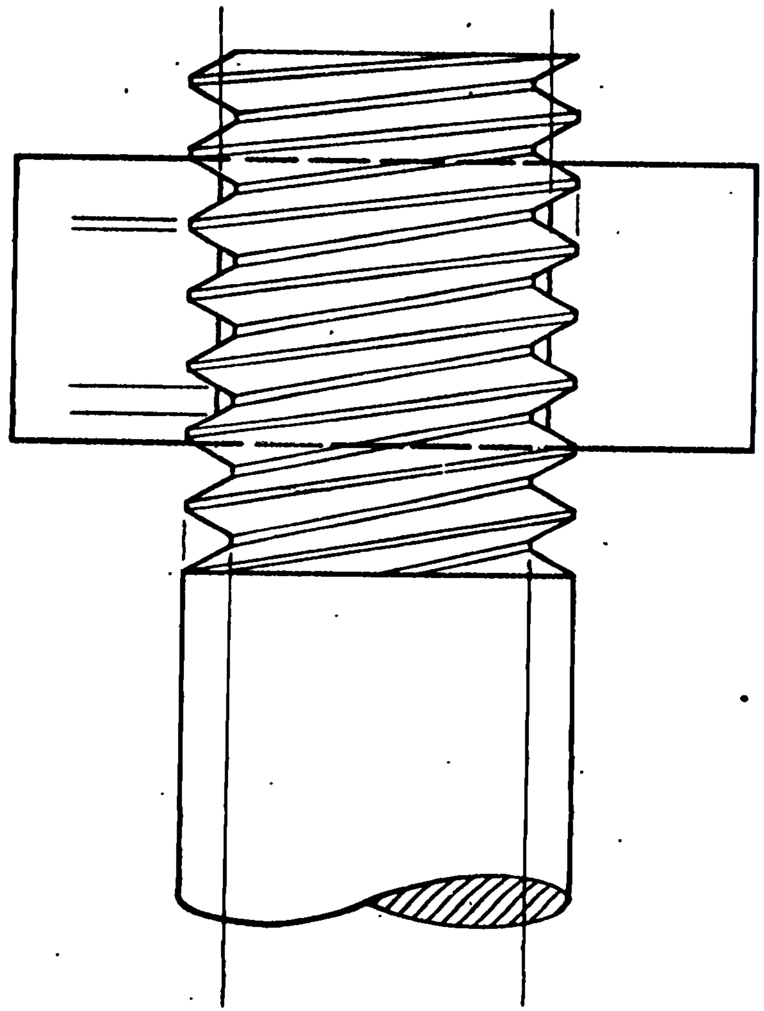
SINGLE DEPTH

30°

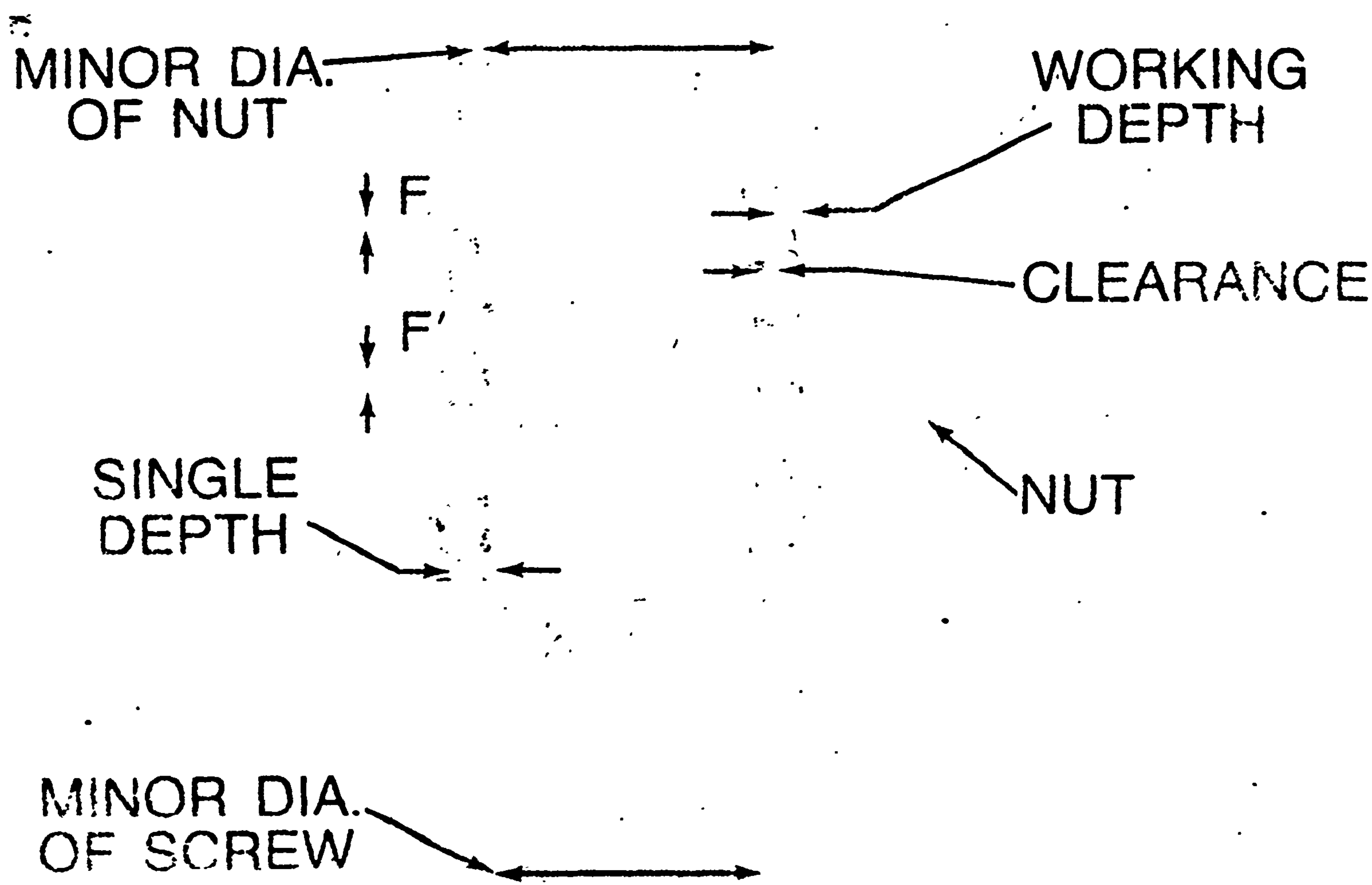
PITCH

THREAD
ANGLE

ROOT

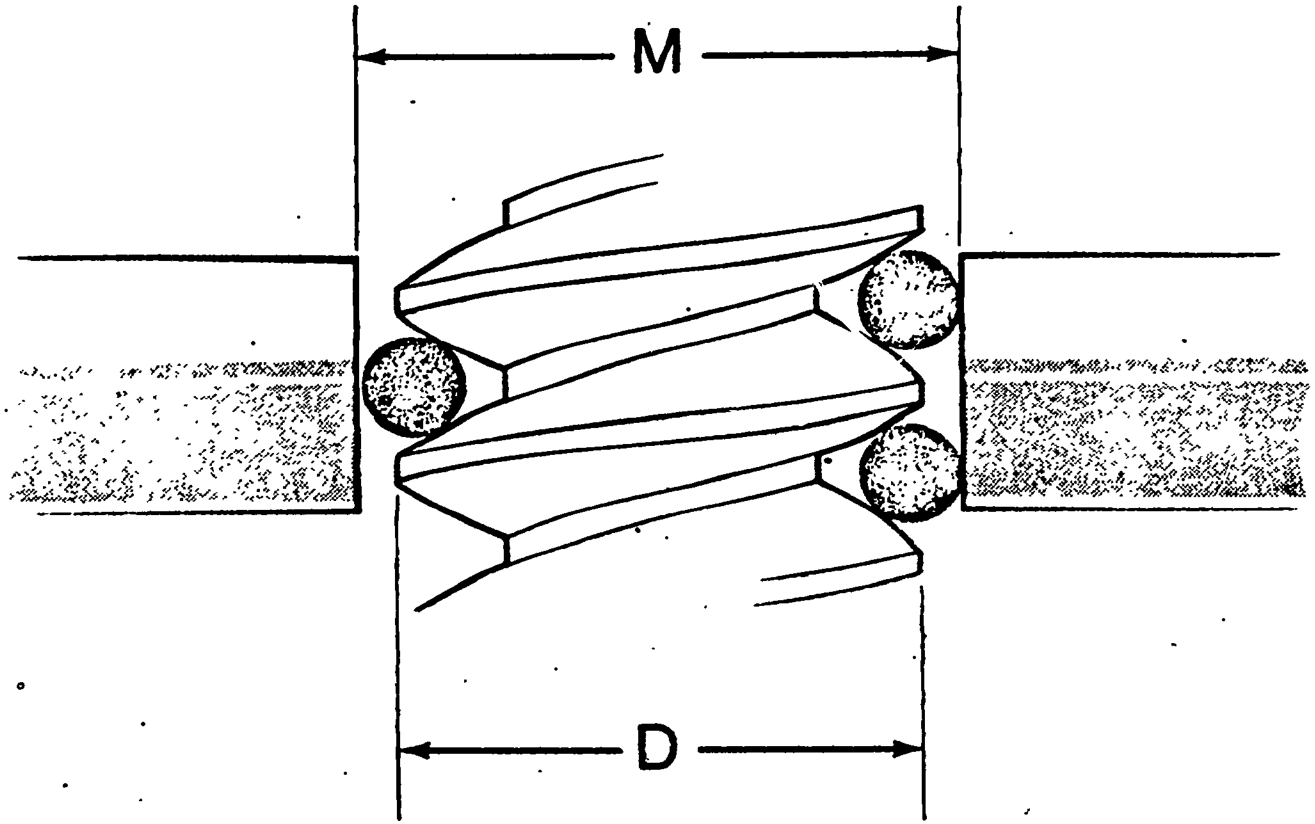


A COMPARISON OF MINOR DIAMETERS

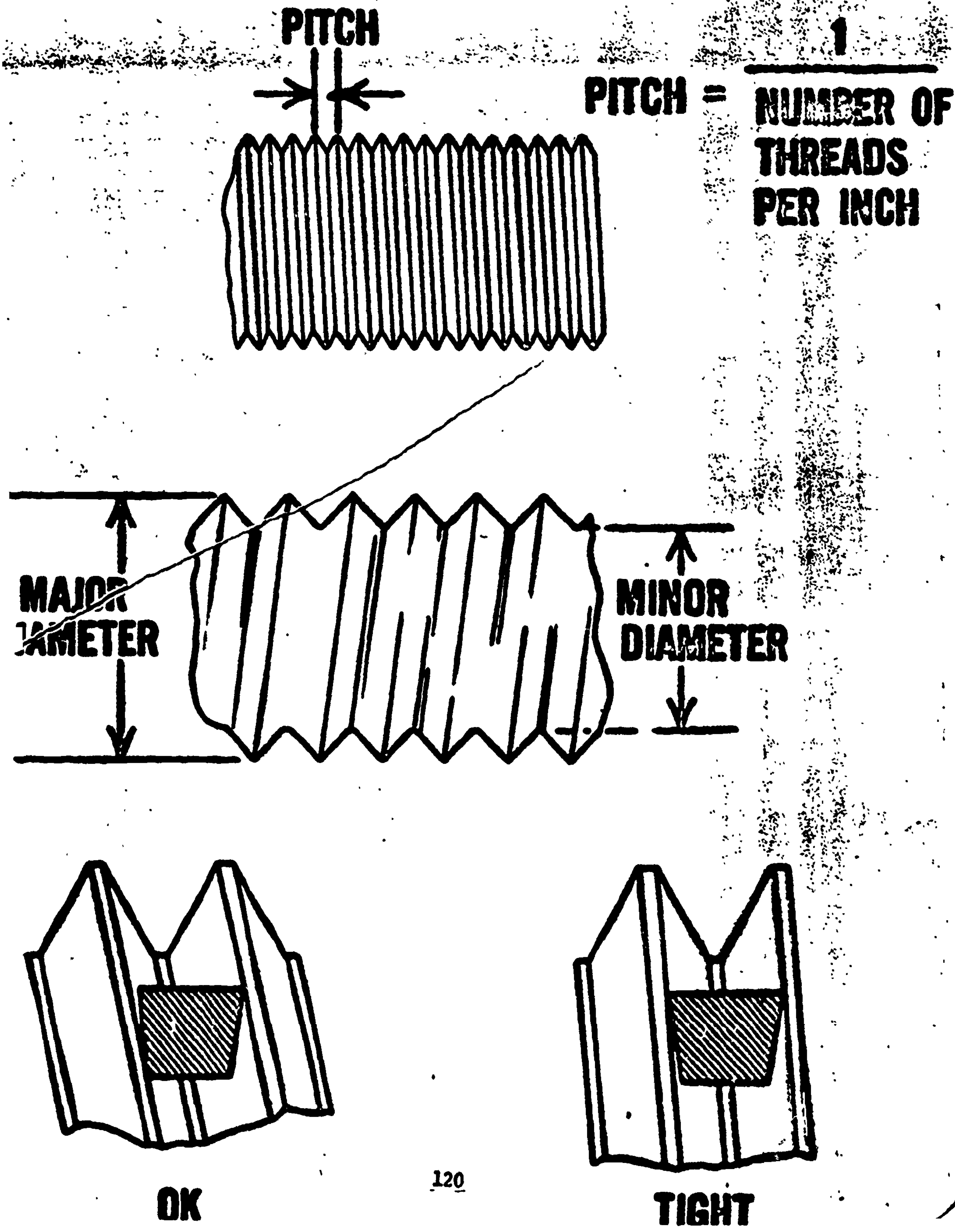

MINOR DIA.
OF NUT
WORKING
DEPTH
F
CLEARANCE
F'
SINGLE
DEPTH
NUT
MINOR DIA.
OF SCREW

D = O.D. OF THREAD

M = MEAS. OVER WIRES



3—WIRE MEASUREMENT OF A THREAD


PITCH
PITCH = 1 / NUMBER OF THREADS PER INCH
MAJOR DIAMETER
MINOR DIAMETER
OK
TIGHT

THREAD FITS

FIT NUMBER

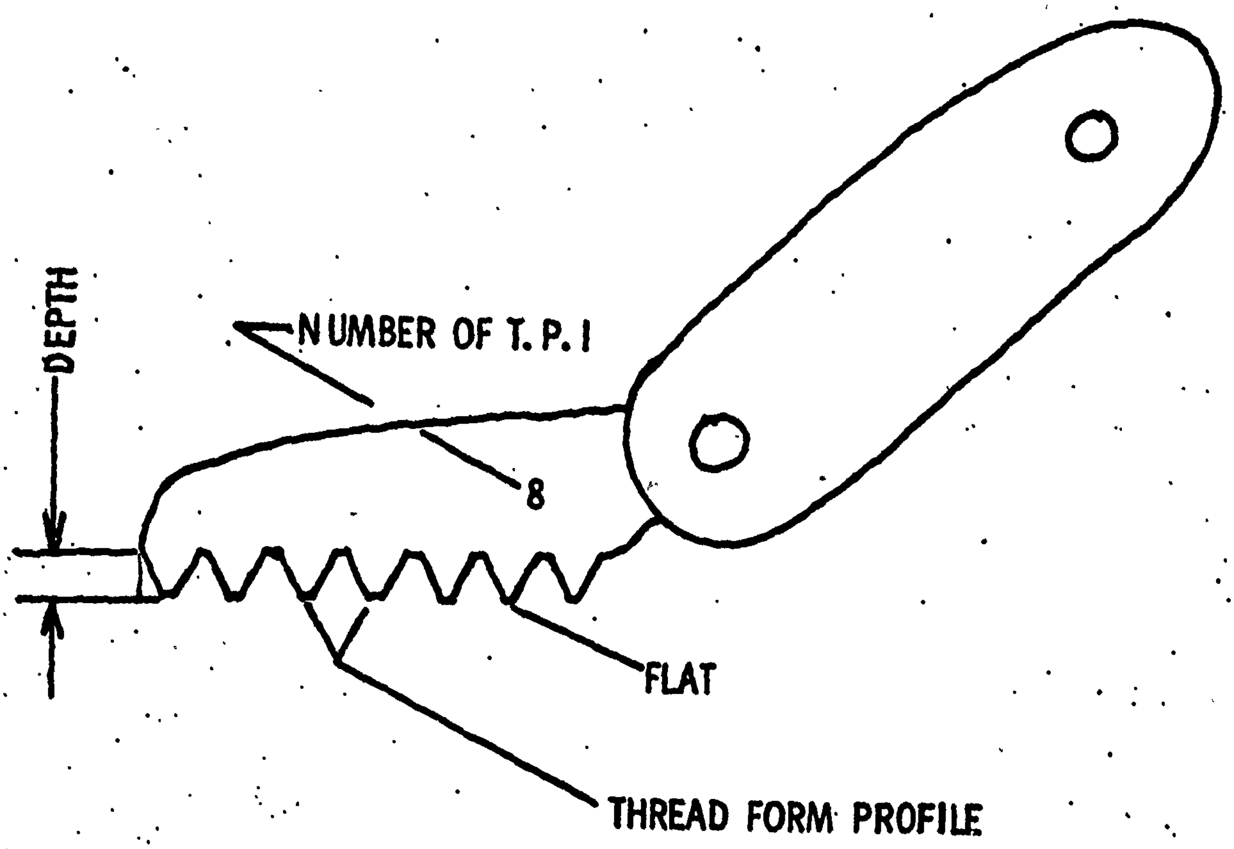
NUMBER 1 LOOSE (SPIN)

NUMBER 2 MEDIUM (GENERAL USE)

NUMBER 3 TIGHT

NUMBER 4 WRENCH

NUMBER 5 SWEAT (HEAT)



THE PITCH GAGE

PITCH

PITCH: DISTANCE FROM TOP OF A THREAD TO THE TOP OF THE NEXT THREAD

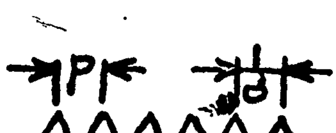
PITCH: $\dfrac{1}{\text{NUMBER OF THREADS PER INCH}}$

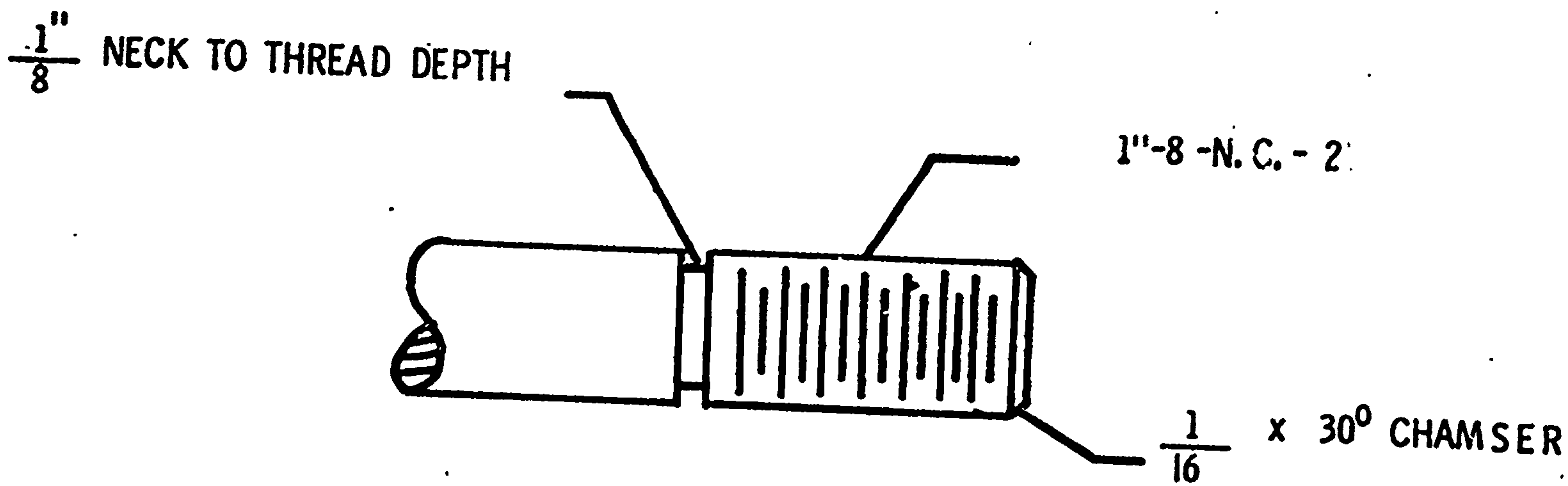
EXAMPLE - 1"-8-N. C. PITCH = $\dfrac{1}{8}$"

DECIMAL EQUIVALENT OF PIT

PITCH = $\dfrac{1"}{8}$

$$
\begin{array}{r}
.125 \\
8\,\overline{)\,1.000} \\
\underline{8} \\
20 \\
\underline{16} \\
40 \\
\underline{40}
\end{array}
$$





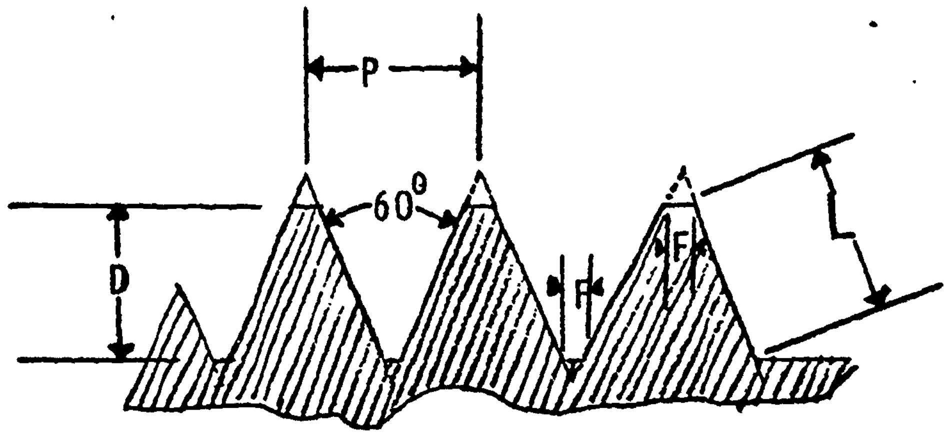
1" - Major Diameter

8 - Threads Per Inch

N. C. - National Course (Series)

2 - Classification Fit

AMERICAN NATIONAL THREAD FORM



PITCH P - NUMBER OF THDS. PER INCH

DEPTH D= .64952 x PITCH

FLAT F = .125 x PITCH

FLAT F = $\frac{P}{8}$

ANGLE = 60 DEGREES

LENGTH L = PITCH x .75 @ 30°

LENGTH L=PITCH X .743 @ 29°

FORMULA TO FIND
PITCH ~ FLAT ~ SINGLE DEPTH ~ DOUBLE DEPTH AND ANGULAR DEPTH

1. P. EQ. PITCH EQ. $\frac{1}{N.T.P.I.}$

2. F. EQ. FLAT EQ .125 X P.

3. D. EQ. SINGLE DEPTH OF THREAD EQ $\frac{.6495}{N.T.P.I.}$

4. D.D. EQ DOUBLE DEPTH OF THREAD EQ 2 X D.

5. A.D. EQ ANGULAR DEPTH EQ SECANT OF ANGLE X D.

6. ANGULAR DEPTH FOR 29° ALSO = $\frac{.743}{N}$

   30° = $\frac{.750}{N}$

FORMULA TO FIND TAP DRILL SIZE

$$DIA - \frac{1}{N.T.P.I.}$$

EXAMPLE

TAP DRILL FOR $\frac{3}{8}$ 16 THD.

T.D.S. = .375 - .062 = .312

OR

$3/8 - 1/16 = 5/16$

FORMULA FOR OUTSIDE DIAMETER OF A NUMBER SIZE SCREW

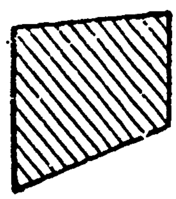
NUMBER x .013 + .060 = OUTSIDE DIAMETER

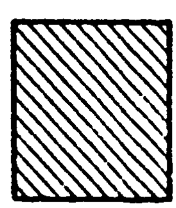
EXAMPLE A - NUMBER 10-32 N.F.

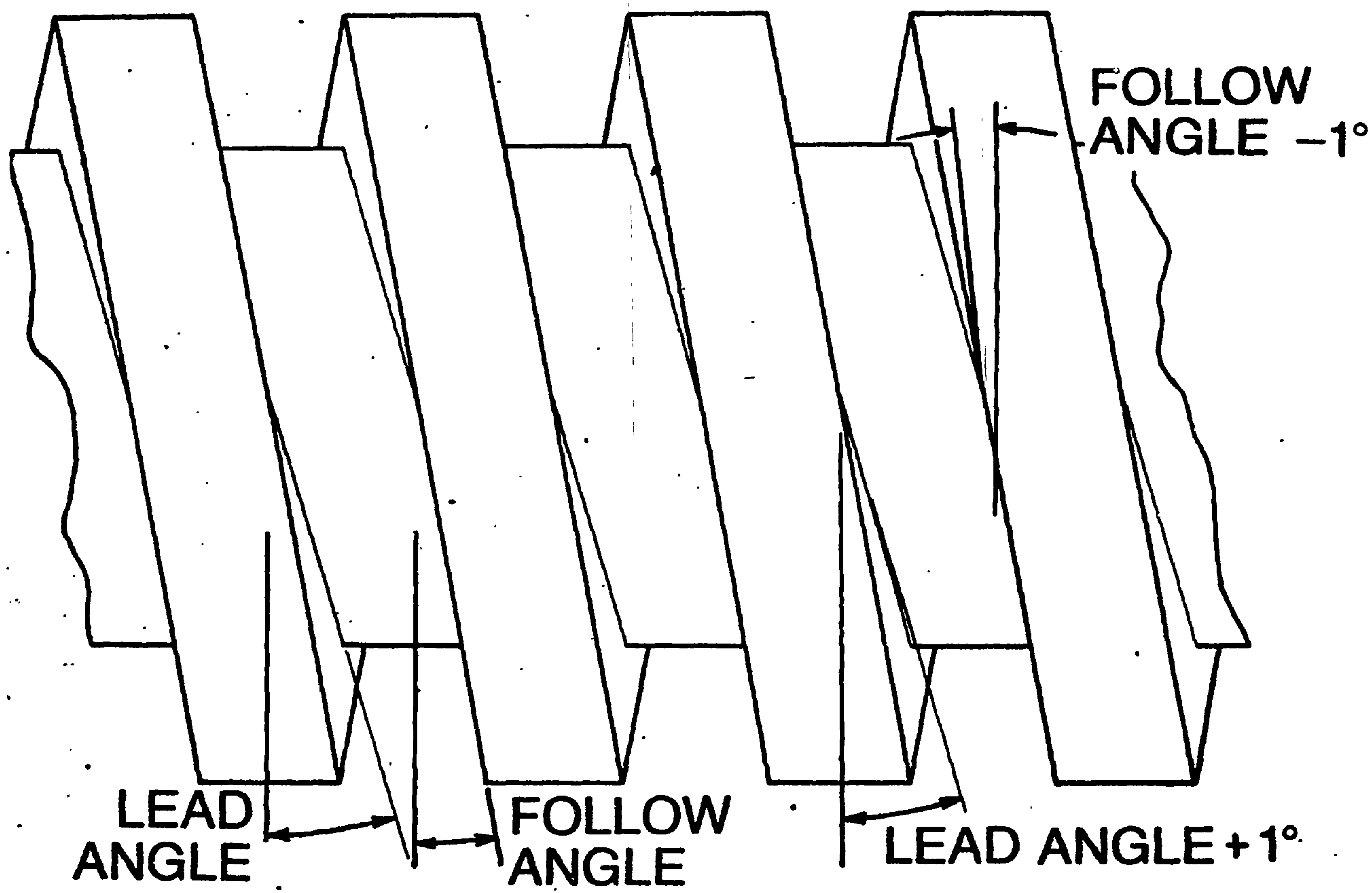
```
 .013 (CONSTANT)
 x 10 NUMBER SIZE
 ----------------
 .130
+.060 (CONSTANT)
 ----------------
 .190 O. D. of A Number 10
                    Screw
```

EXAMPLE B - NUMBER 8-40 N.F.

```
 .013
 x 8 NUMBER SIZE
 ----------------
 .104
+.060
 ----------------
 .164 O. D. of A Number 8
                   Screw
```





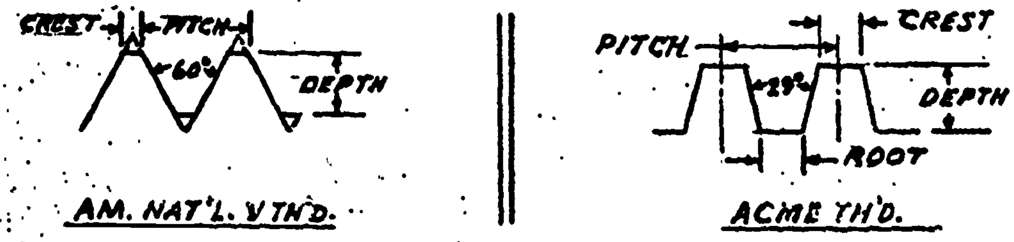


LEADING AND FOLLOWING ANGLES OF A SQUARE THREAD TOOLBIT

SHOP MATH

CALCULATION OF ACME THREAD FORM

THE ACME THREAD IS GENERALLY USED TO TRANSMIT POWER AND MOTION AND DIFFERS FROM THE CONVENTIONAL V THREAD IN THAT IT HAS AN INCLUDED THREAD ANGLE OF 29° COMPARED TO THE V THREAD 60° ANGLE. THE ILLUSTRATION BELOW SHOWS DIFFERENCES IN THREAD FORM OF AMERICAN NATIONAL AND ACME THREADS.



FORMULAS FOR ACME THREAD PROBLEMS

PITCH = $\frac{1}{\text{NO. THDS. PER IN.}}$

DEPTH = $\frac{\text{PITCH}}{2}$ + .010"

WIDTH OF CREST = 0.3707 X PITCH

WIDTH OF ROOT = 0.3707 X PITCH − 0.0052"

PROBLEM: CALCULATE THE PITCH, DEPTH, WIDTH OF CREST, AND WIDTH OF ROOT OF 8 THREADS PER INCH ACME THREAD FORM.

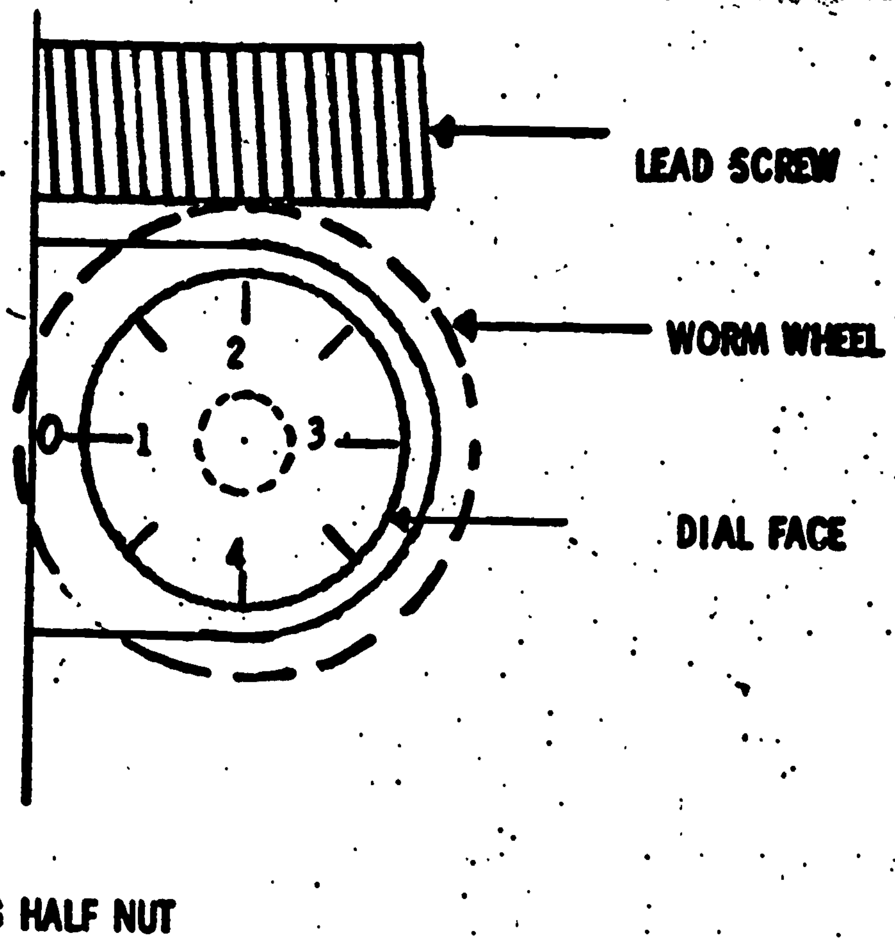
PITCH = 1 ÷ 8 = 0.125"

DEPTH = 0.125" ÷ 2 = 0.0625 + 0.010" = 0.0725"

WIDTH OF CREST = 0.3707 X 0.125 = 0.0463"

WIDTH OF ROOT = 0.3707 X 0.125 − 0.0052" = 0.0411"

ARRANGEMENT OF THREAD CHASING DISH



EVEN THREADS ANY LINE

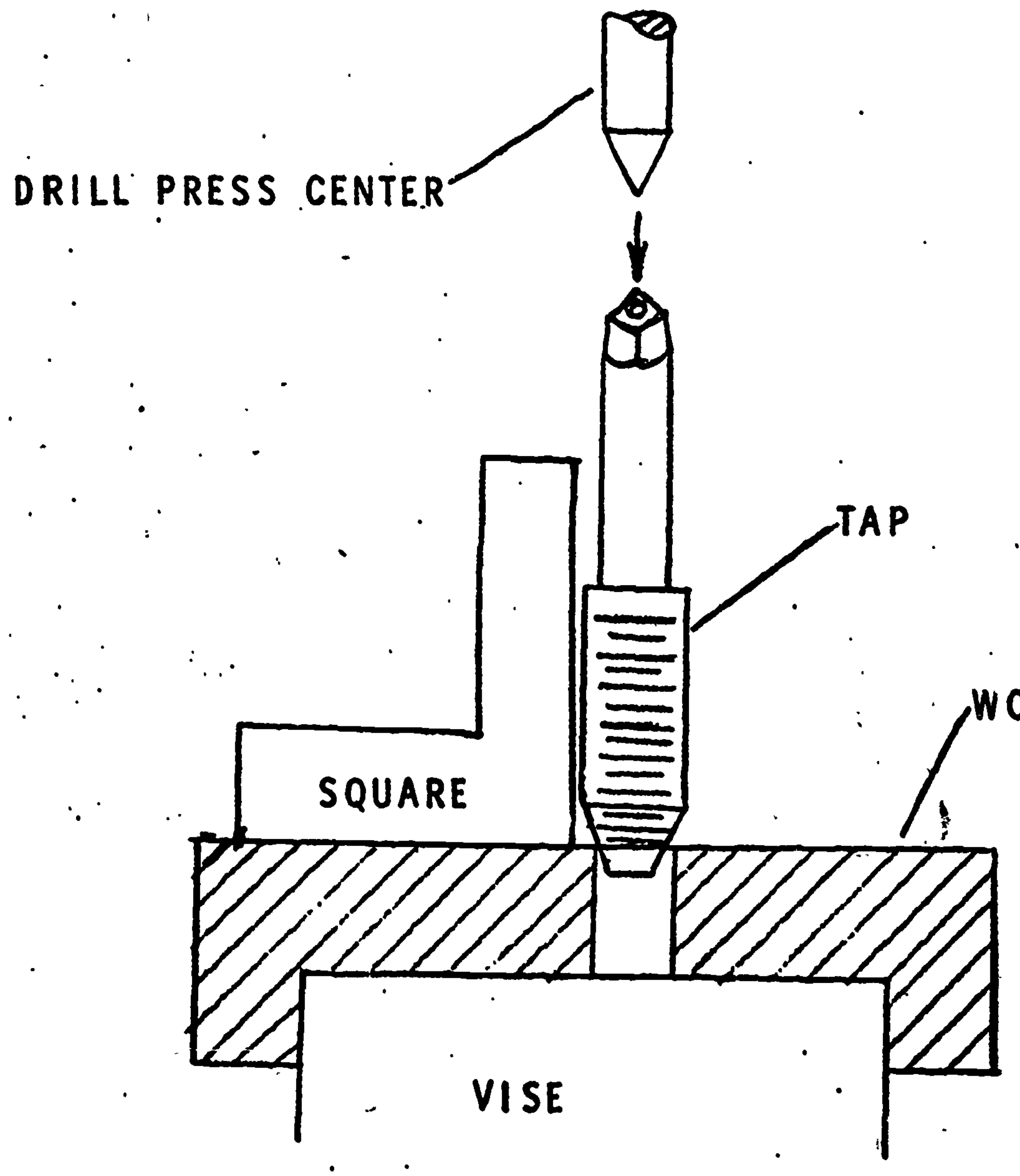
ODD THREADS ANY NUMBERED LINE.

HALF THREADS ANY ODD NUMBERED LINE.

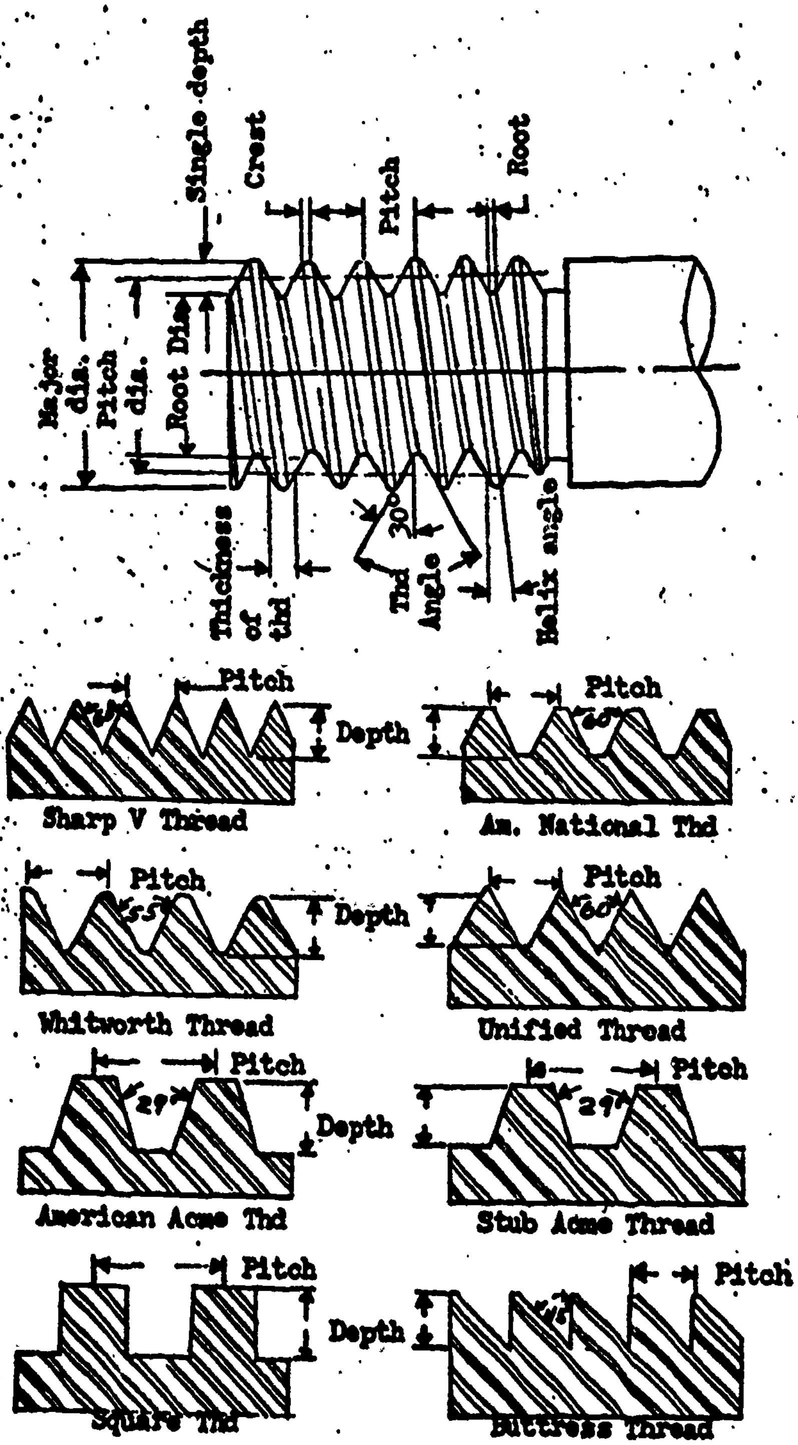
QUARTER THREADS SAME LINE

LUBRICANTS FOR TAP AND DIE OPERATIONS

MILD STEELS - (H.R. or C.R.)	SOLUBLE or LARD OIL
TOOL STEEL - (HISPEED or CARBON)	SULPHUR BASE or MINERAL LARD
MALLEABLE IRON	SOLUBLE
CAST IRON	DRY
BRASS or BRONZE	DRY or SOLUBLE
COPPER	SOLUBLE
ALUMINUM	SOLUBLE



TAP ALIGNMENT



THREAD NOMENCLATURE

TITLE: SCREW THREAD PROBLEMS

UNIT: SCREW THREADS

OCCUPTION: MACHINIST

OBJECTIVE: To give the student practice in solving screw thread problems.

REFERENCE: 1. Axelrod, Aron, <u>Machine Shop Mathematics</u>. New Yoark: McGraw-Hill Book Co., Inc.

2. Information Sheet. <u>Screw Thread Formulas</u>

DIRECTIONS: Read the above references, and using the Information Sheet as a guide work the following problems.

PROBLEMS: 1. Find the single depth of the following screw threads.

a. $\frac{1}{2}$-20-NF

b. $\frac{1}{4}$-28-NF

c. $\frac{1}{4}$-20-NC

d. 5/8-11-NC

e. 3/4-10-NC

2 . Find the double depth of the following screw threads.

a. $\frac{1}{2}$-20-NC

b. 10-32-NF

c. 5/16-18-NC

d. 3/4-16-NF

e. 7/8-9-NC

3. What would be the width of the crest of the following screw threads?

a. $\frac{1}{4}$-20-NC

b. $\frac{1}{2}$-20-NF

c. 3/8-16-NC

d. 3/4-10-NC

e. 1-8-NC

4. What is the pitch of the following screw threads?

a. $\frac{1}{4}$-28-NF

b. 3/8-24-NC

c. $\frac{1}{2}$-20-NF

d. 5/8-11-NC

e. 3/4-10-NC

5. What would be the tap drill size for the following screw threads?

a. $\frac{1}{4}$-20-NC

b. 3/8-16-NC

c. $\frac{1}{2}$-20-NF

d. $\frac{1}{2}$-13-NC

e. 5/8-18-NF

ASSIGNEMTN SHEET

TITLE: MILLING MACHINES AND MILLING

UNIT: MILLING MACHINE AND MILLING

OCCUPATION: MACHINEST

OBJECTIVE: To familiarize the student with the types of milling machines and milling operations.

REFERENCE: Anderson-Tatro. Shop Theory. New York: Mc-Graw Hill Book Co., Inc. Chapter 10, pages 234-278.

DIRECTIONS: Read the above refbrence and answer the following questions.

QUESTIONS:

1. What are the principal parts of a milling machine and what is the function of each?

2. How does a universal differ from a plain milling machine?

3. What are the three general types of milling machines?

4. How is feed expressed on a milling machine?

5. What are the feed movements on a knee and column milling machine?

6. What are the advantages of a vertical milling machine?

7. How is the speed expressbd on a milling machine?

8. How is the size of a milling machine determined?

9. What is the function of the following milling machine attachments?

   (A) Rotary table
   (B) Rack Milling
   (C) Arbor
   (D) Slotting
   (E) Index head
   (F) Adaptor

10. List and give the use of six types of milling cutters.

11. Explain how cutters are mounted on milling machines.

12. List and explain the ways of mounting work on a millingmachine.

13. What is up and down milling?

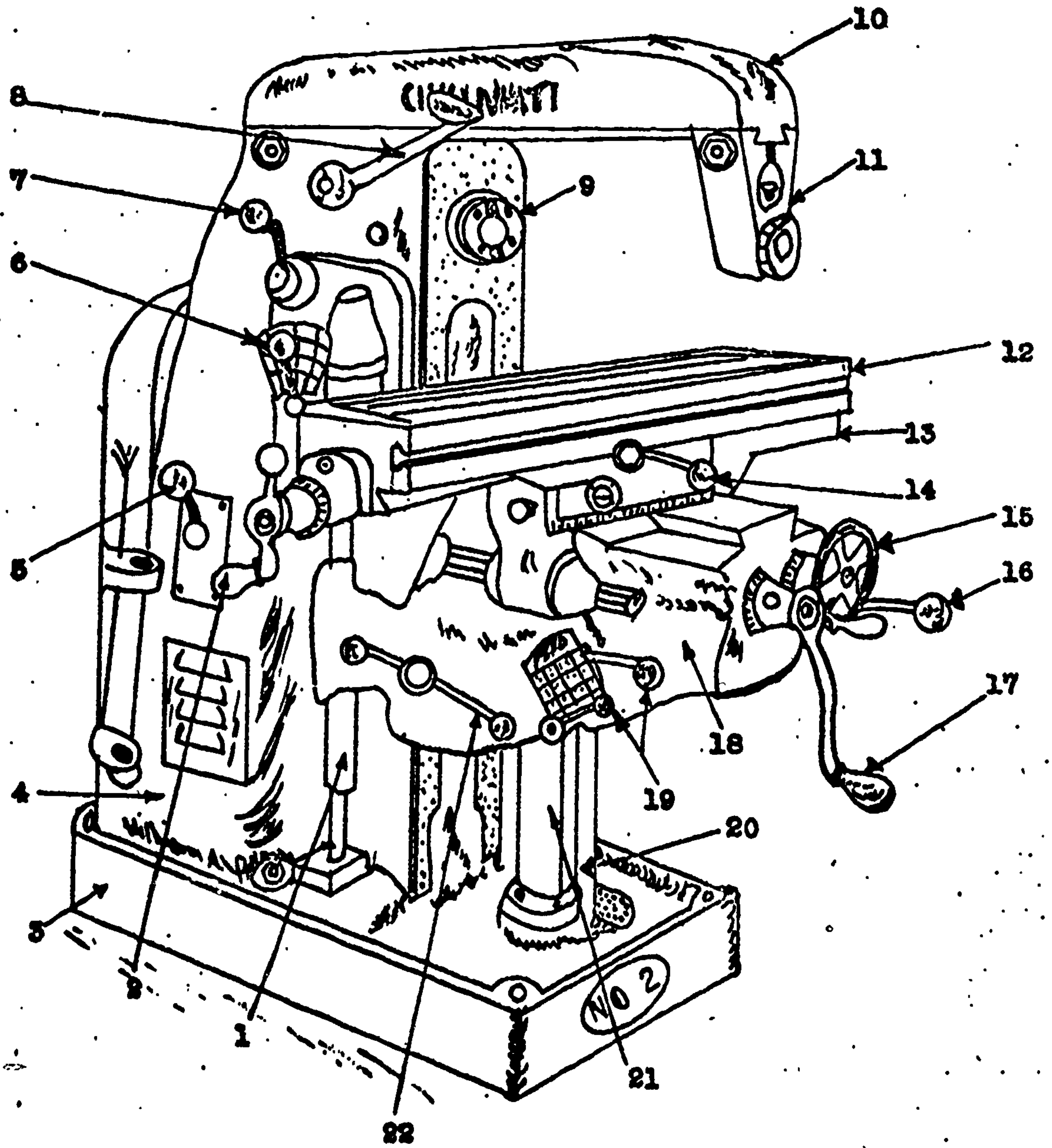
14. Explain the difference between plain and rapid indexing.

15. How do you distinguish between a right hand and a left hand end mill?

CINCINNATI UNIVERSAL MILLING MACHINE

COLUMN AND KNEE TYPE

NOMENCLATURE



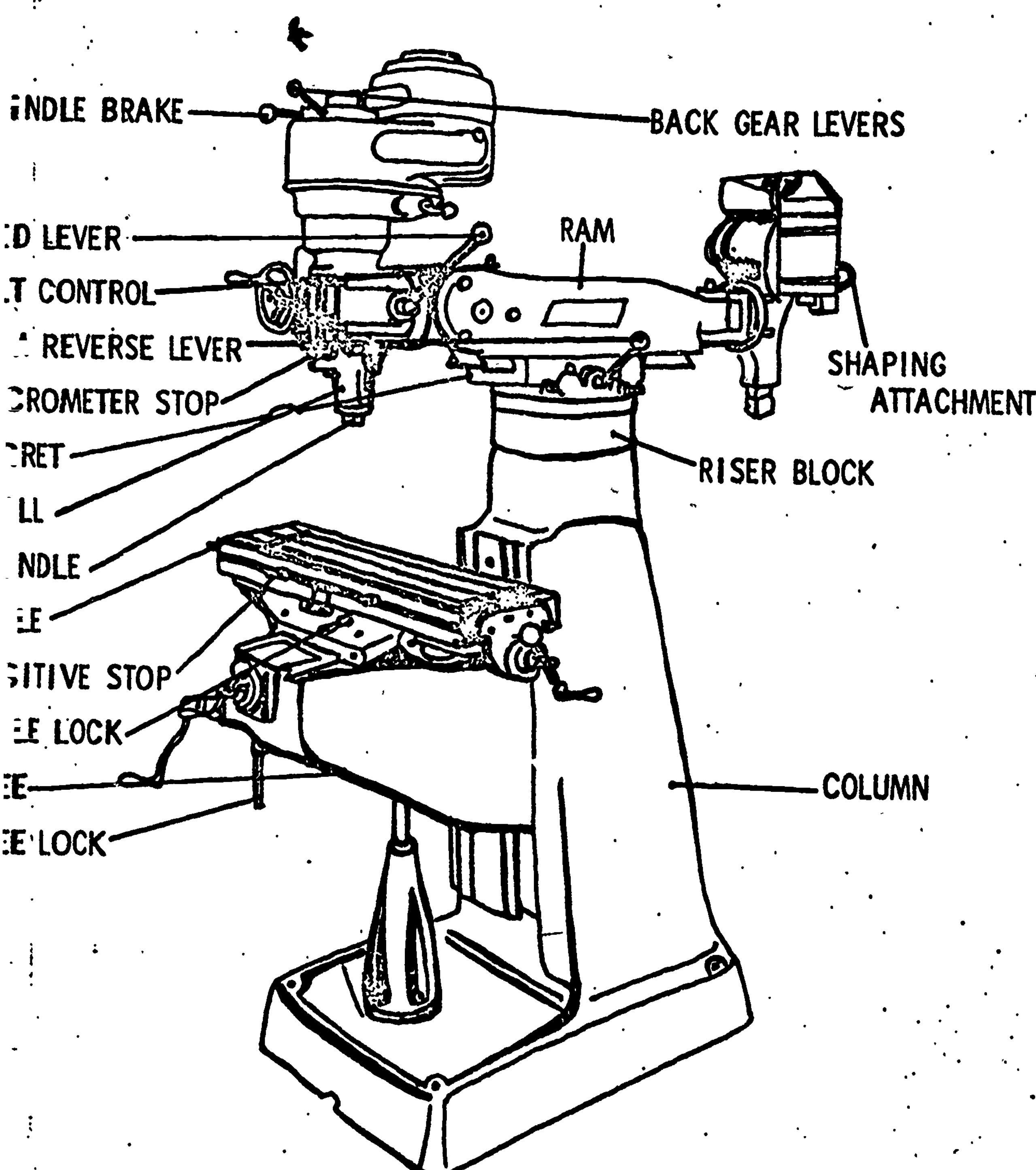
CINCINNATI UNIVERSAL MILLING MACHINE

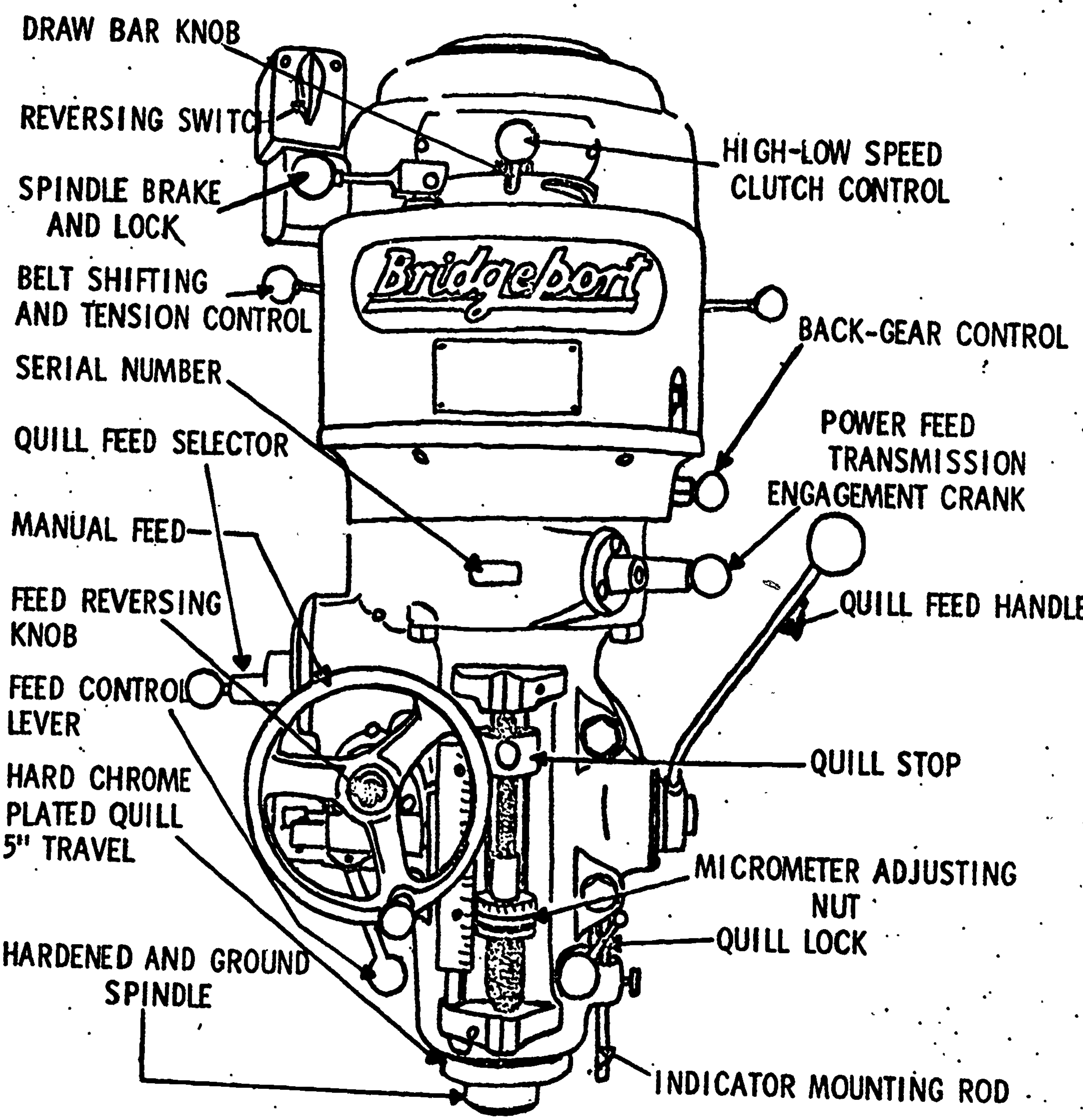
COLUMN AND KNEE TYPE

NOMENCLATURE

1. FEEDS DRIVE SHAFT
2. LONGITUDINAL HAND FEED LEVER
3. BASE
4. COLUMN
5. SPINDLE REVERSING LEVER
6. LOWER SPEED CHANGE LEVER
7. UPPER SPEED CHANGE LEVER
8. STARTING LEVER
9. SPINDLE No. 50 TAPER Inside
10. OVERARM
11. ARBOR SUPPORT
12. TABLE
13. SADDLE
14. TABLE FEED REVERSING LEVER
15. TRANSVERSE HAND WHEEL
16. RAPID TRAVERSE
17. VERTICAL HAND FEED LEVER
18. KNEE
19. FEED CHANGE LEVERS
20. COOLANT DRAIN TO BASE
21. ELEVATING KNEE SCREW
22. FEEDS REVERSING LEVER

Best Copy Available

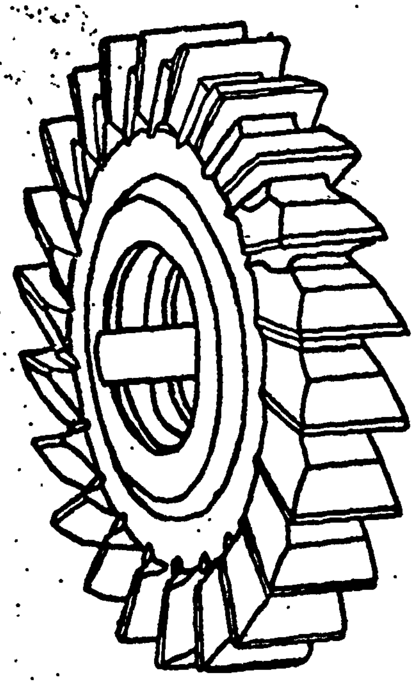


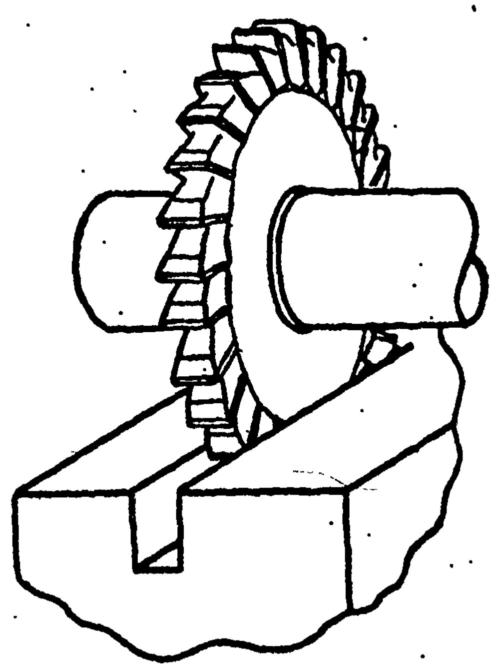

DRAW BAR KNOB
REVERSING SWITCH
SPINDLE BRAKE AND LOCK
BELT SHIFTING AND TENSION CONTROL
SERIAL NUMBER
QUILL FEED SELECTOR
MANUAL FEED
FEED REVERSING KNOB
FEED CONTROL LEVER
HARD CHROME PLATED QUILL 5" TRAVEL
HARDENED AND GROUND SPINDLE
Bridgeport
HIGH-LOW SPEED CLUTCH CONTROL
BACK-GEAR CONTROL
POWER FEED TRANSMISSION ENGAGEMENT CRANK
QUILL FEED HANDLE
QUILL STOP
MICROMETER ADJUSTING NUT
QUILL LOCK
INDICATOR MOUNTING ROD

M 10 112

#1

SIDE MILLING CUTTER





SIDE MILLING IS THE MACHINING OF A VERTICAL SURFACE ON THE SIDE OF THE WORK WITH A SIDE-MILLING CUTTER.

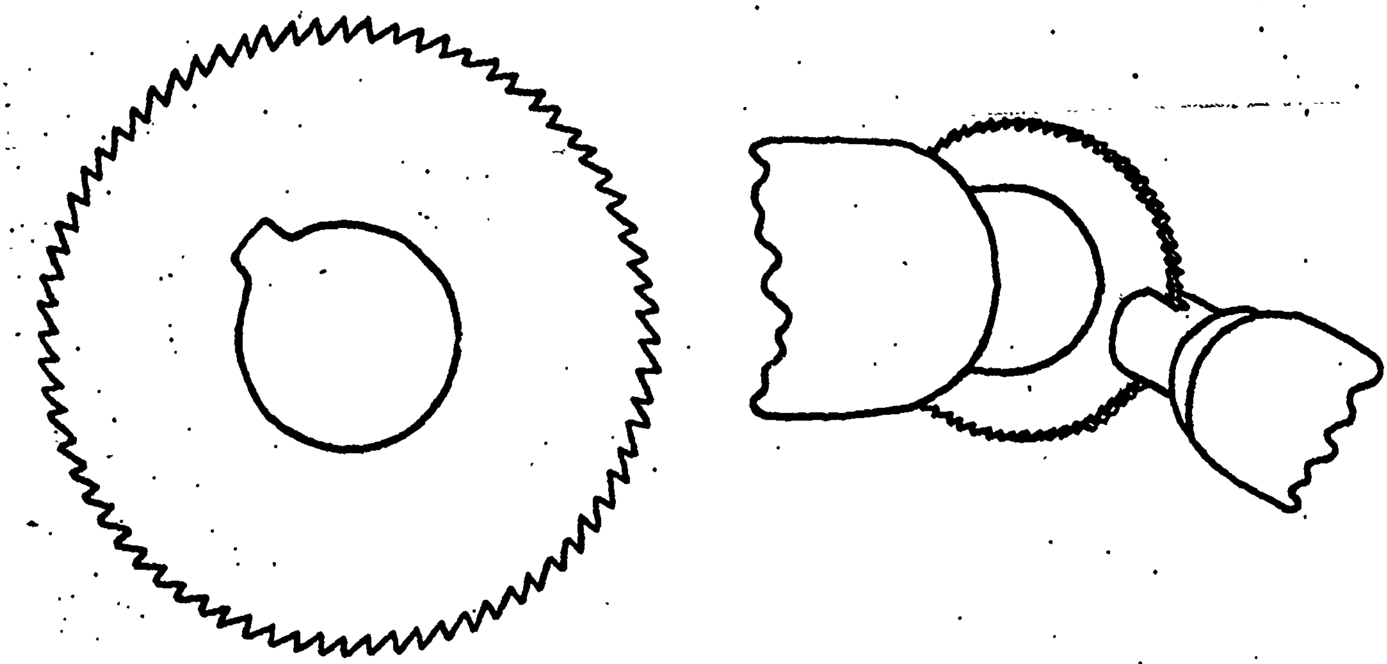
SIDE MILLING CUTTERS HAVE TEETH ON BOTH SIDES AND THEIR PERIPHERY.

CAN BE OBTAINED IN WIDTHS TO 1" WITH STRAIGHT OR HELICAL TEETH.

USED FREQUENTLY FOR MILLING SLOTS, BUT CAN, ALSO, BE USED FOR STRADDLE MILLS BY GANGING.

#2

PLAIN SCREW SLOTTING CUTTER



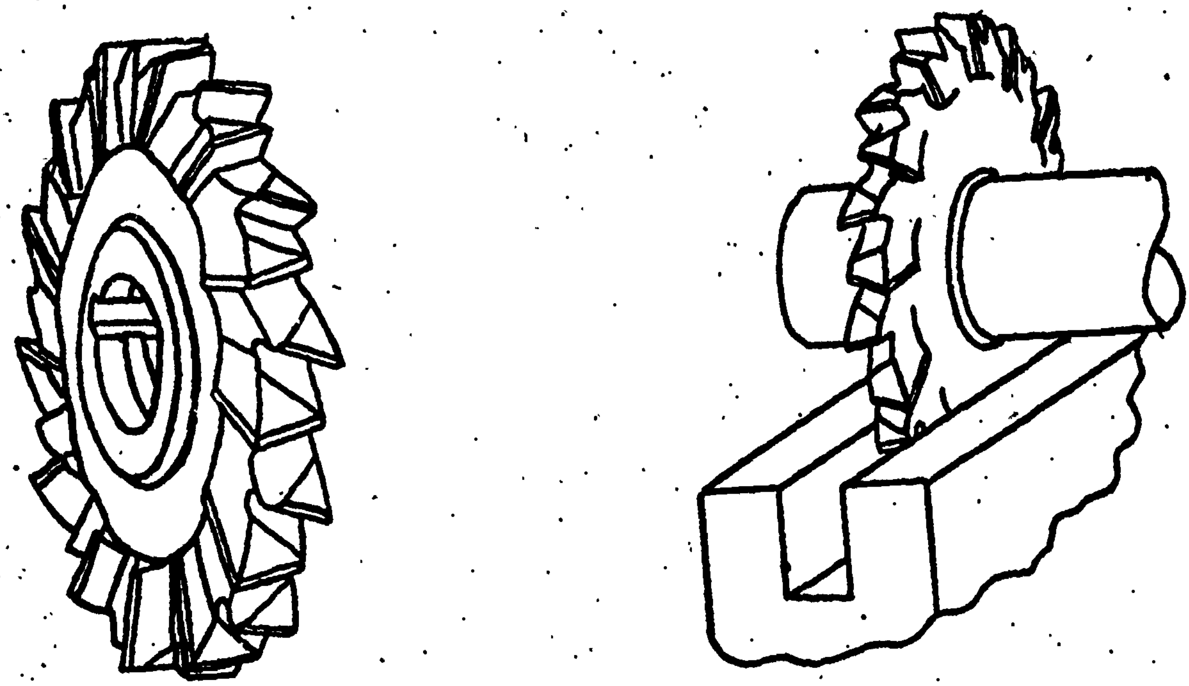
SLOTTING IS THE CUTTING OF GROOVES WHICH HAVE VERTICAL SIDES IN EITHER FLAT OR CYLINDRICAL WORK BY MEANS OF THE APPROPRIATE CUTTER.

PLAIN SCREW SLOTTING CUTTERS HAVE FINE PITCHED TEETH ON THE PERIPHERY AND HAVE THE SIDES DISHED TO PROVIDE CLEARANCE.

USED FOR SLOTTING SCREW HEADS, SLITTING TUBING, THIN SHEET METAL, AND OTHER SHALLOW OPERATIONS.

#3

STAGGERED TOOTH SIDE MILLING CUTTER



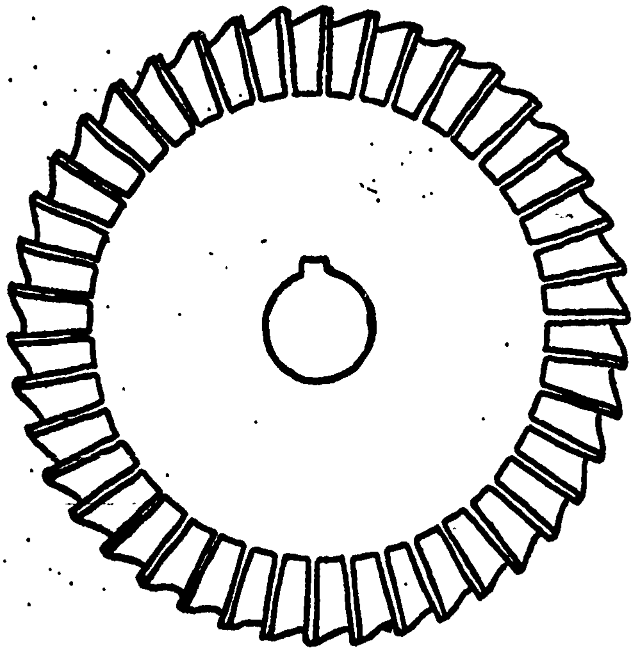
STAGGERED TOOTH SIDE MILLING CUTTERS ARE DESIGNED ESPECIALLY FOR DEEP SLOTTING, KEYWAYS AND HEAVY DUTY MILLING.

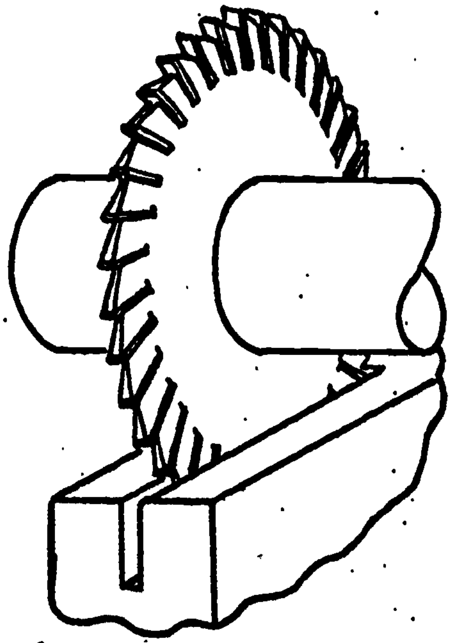
THEY ARE MADE WITH ALTERNATE SIDE TEETH, WHICH HAVE ALTERNATE RIGHT AND LEFT HAND HELIX. IT HAS A SHEARING ACTION, WHICH IS DESIRABLE FOR GOOD CUTTING AND FINISHES. REQUIRES LESS POWER DUE TO SHEARING ACTION OF THE HELIX ON THE TEETH.

8.1

#4

SIDE TOOTH METAL SLITTING SAW





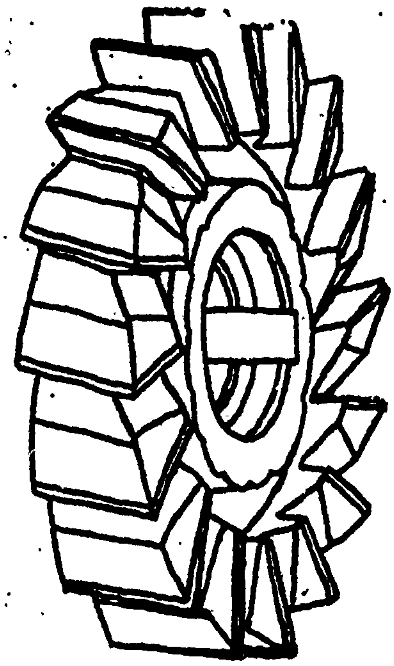
SLITTING IS THE OPERATION PERFORMED WHEN THE CUTTING OF A GROOVE INTO THE WORK OR CUTTING OFF AS IN A PARTING OPERATION.

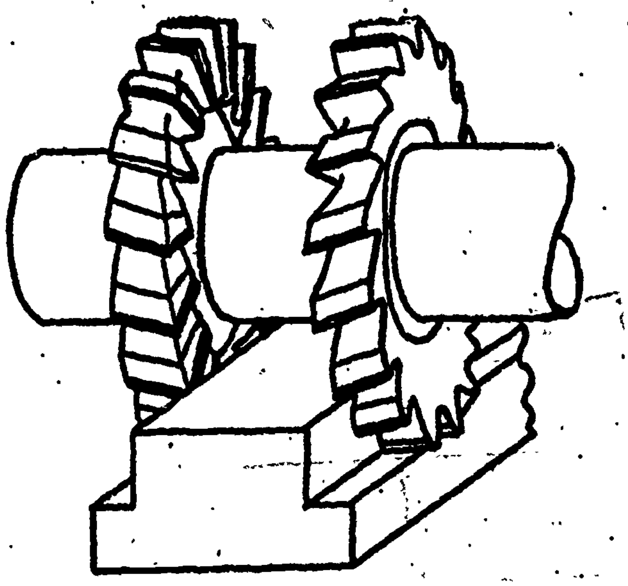
THE SIDE TEETH GIVE GREATER CHIP CLEARANCE AND A WIDER SPACING OF THE TOOTH. IT IS RELEIVED TOWARD THE CENTER TO ALLOW FOR CLEARANCE.

USED FROM NORMAL TO DEEP SLOTTING AND CUTTING OFF OPERATIONS.

#5

HALF SIDE MILLING CUTTER



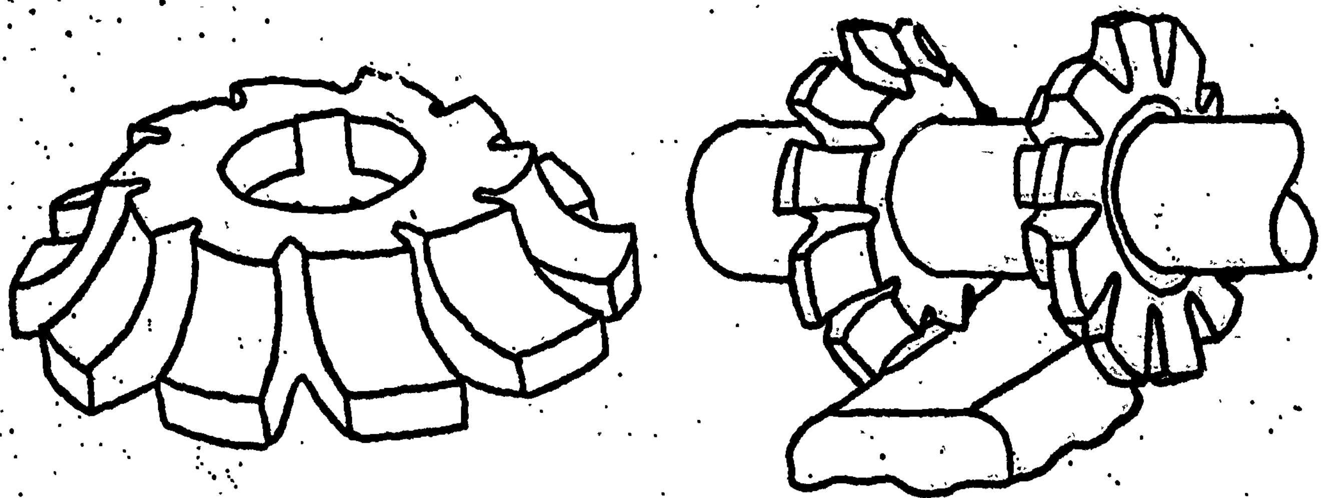


HALF SIDE MILLING CUTTERS HAVE TEETH ON ONE SIDE AND THE PERIPHERY.

MADE WITH LEFT AND RIGHT HAND HELICAL TEETH.

USED PRINCIPALLY TO STRADDLE MILL WHERE YOU NEED A SHOULDER AND A FLAT IN ONE OPERATION. THEY CAN BE USED SINGLEY OR GANGED IN PAIRS. EXCELLENT FOR HEAVY CUTS AND WHERE CLOSE TOLERANCES ARE NEEDED BETWEEN SHOULDERS.

CORNER ROUNDING CUTTER



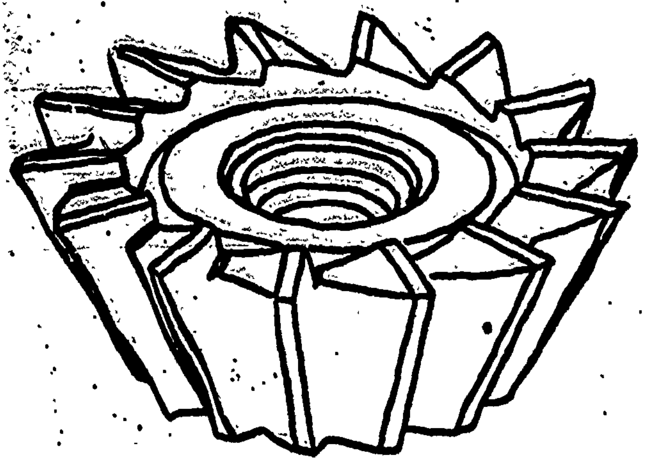
CORNER ROUNDING CUTTERS ARE USED TO PRODUCE CONVEX SURFACES OF CIRCULAR CONTOUR EQUAL TO A QUARTER CIRCLE OF LESS.

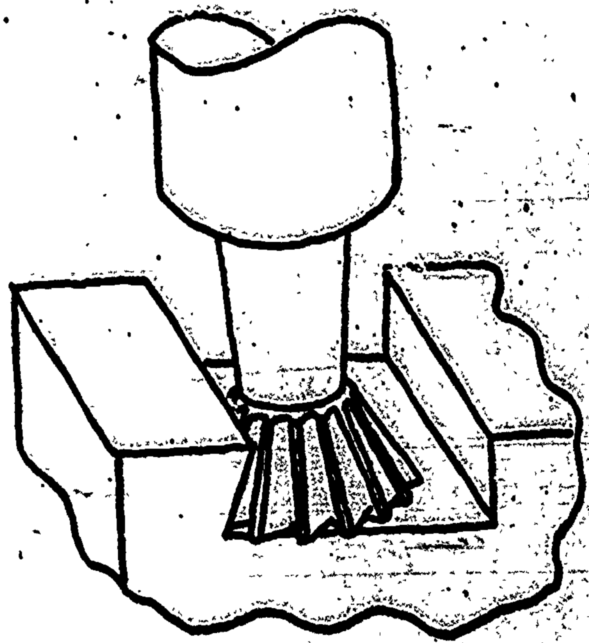
THEY ARE ONE OF THE MANY FORM CUTTERS AVAILABLE, SUCH AS, CONVEX, CONCAVE, FLUTING, AND DOUBLE ANGLE RADIUS CUTTERS FOR MILLING THE MANY DIFFERENT FORMS.

CAN BE USED SINGLEY OR GANGED. THEIR FORM CAN BE RETAINED BECAUSE THEY ARE FORM RELEIVED AND ARE SHARPENED ON THEIR FACES ONLY.

#8

SINGLE ANGLE MILLING CUTTERS WITH THREADED HOLE





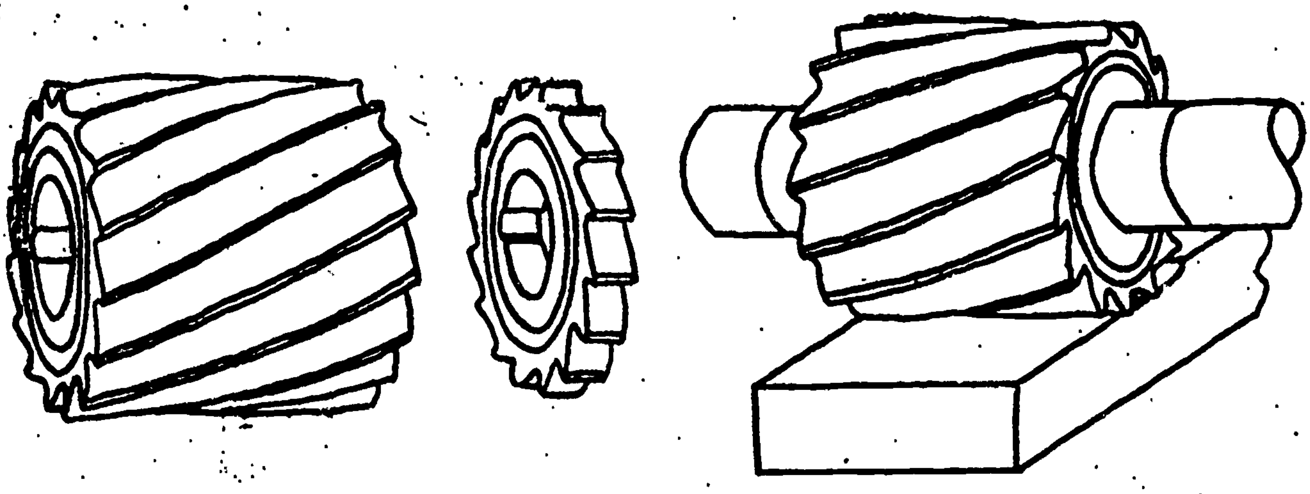
ANGULAR MILLING IS THE MACHINING OF SURFACES AT ANGLES OTHER THAN 90° TO THE ARBOR.

THIS CUTTER IS FURNISHED IN ALL ANGLES. MOST USUALLY A 60° IS USED PRINCIPALLY FOR MILLING DOVE TAILS. IT IS DRIVEN WITH A THREADED END MILLING MACHINE ARBOR.

CARE MUST BE TAKEN AS TO DIRECTION OF CUT IN RELATION TO THE THREAD, WHETHER A RIGHT HAND OR LEFT HAND THREADED ARBOR IS USED.

#9

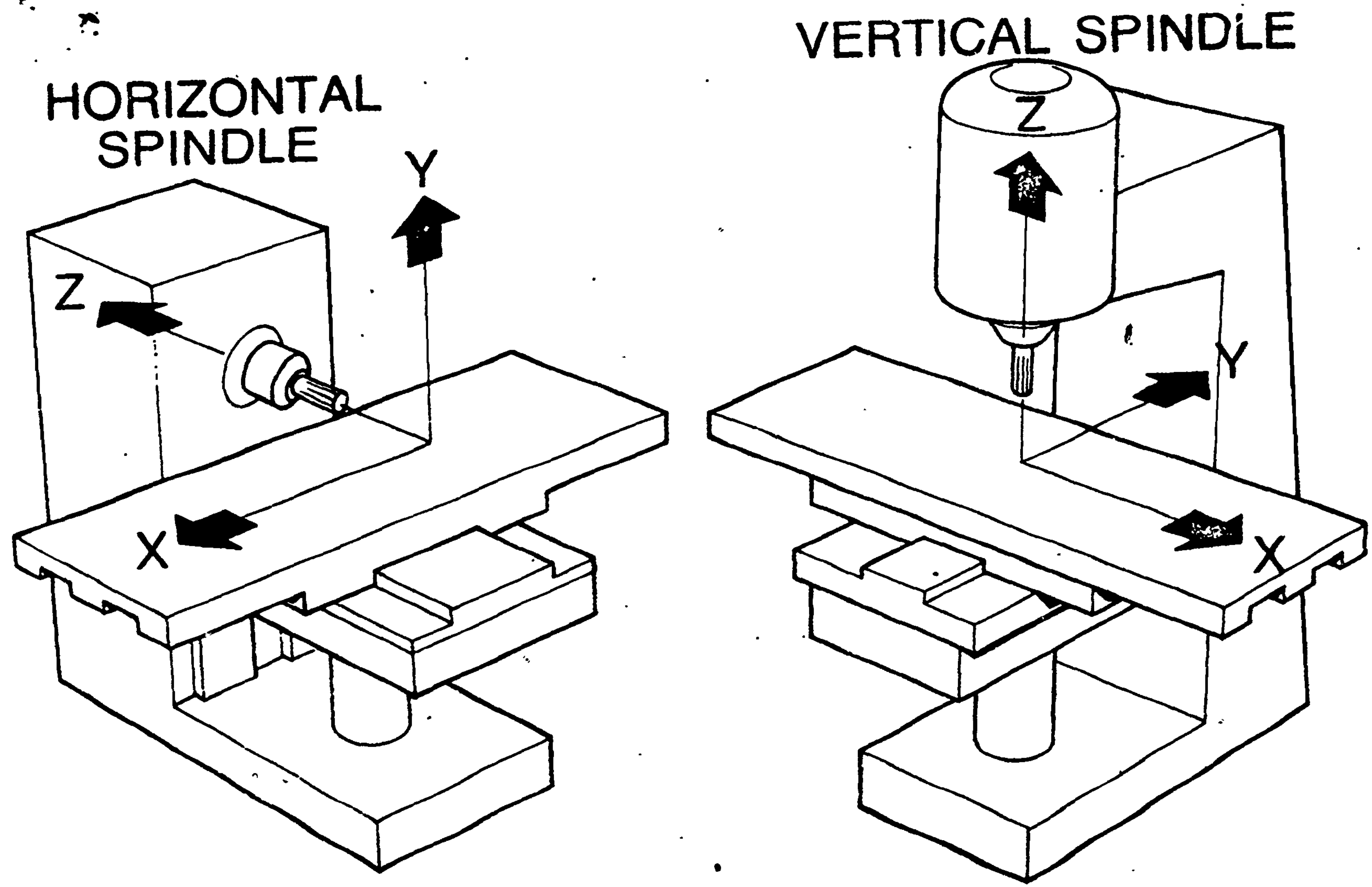
PLAIN MILLING CUTTER
HELICAL SLAB



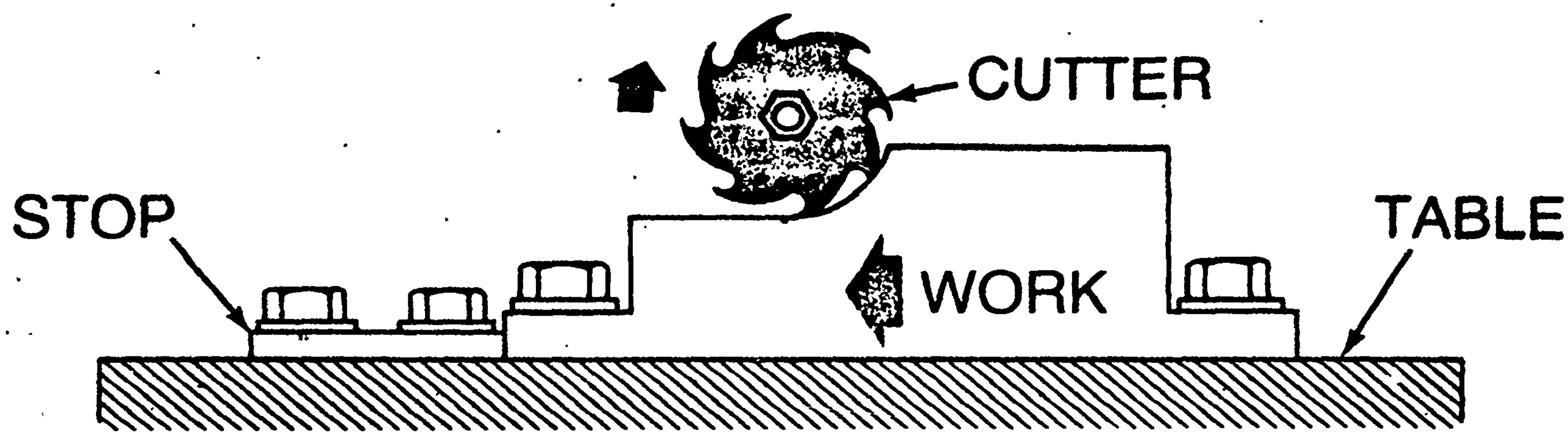
PLAIN MILLING IS THE PRODUCTION OF FLAT SURFACES WITH A PLAIN MILLING, WITH EITHER STRAIGHT OR HELICAL TEETH.

SLAB MILLING OPERATION PRODUCES A FINE FINISH WITH LITTLE OR NO CHATTER WHEN USING A HELICAL TOOTH MILLING CUTTER. THEY CUT ON THE PERIPHERY ONLY AND HAVE WIDTHS FROM 1/8" TO 6" WITH FINE OR COARSE PITCH TEETH.

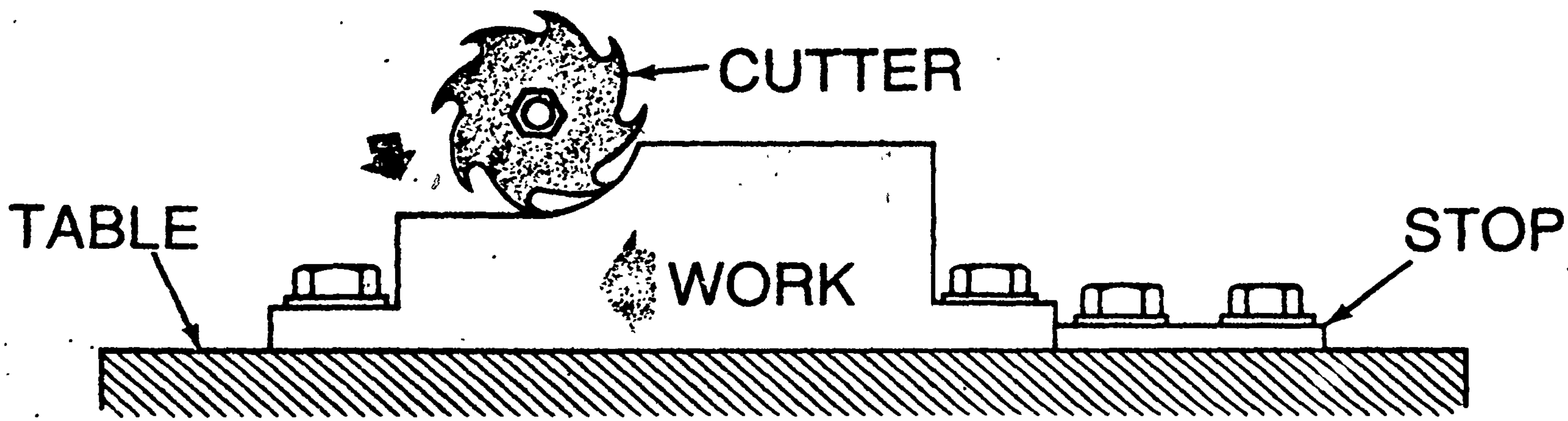
IT IS PRIMARILY USED ON CUTS WHERE THE CUT IS LESS THAN THE FACE WIDTH OF THE CUTTER.



AXES OF MACHINE TOOL MOVEMENTS

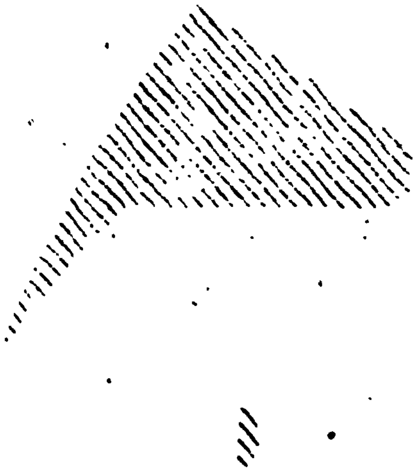


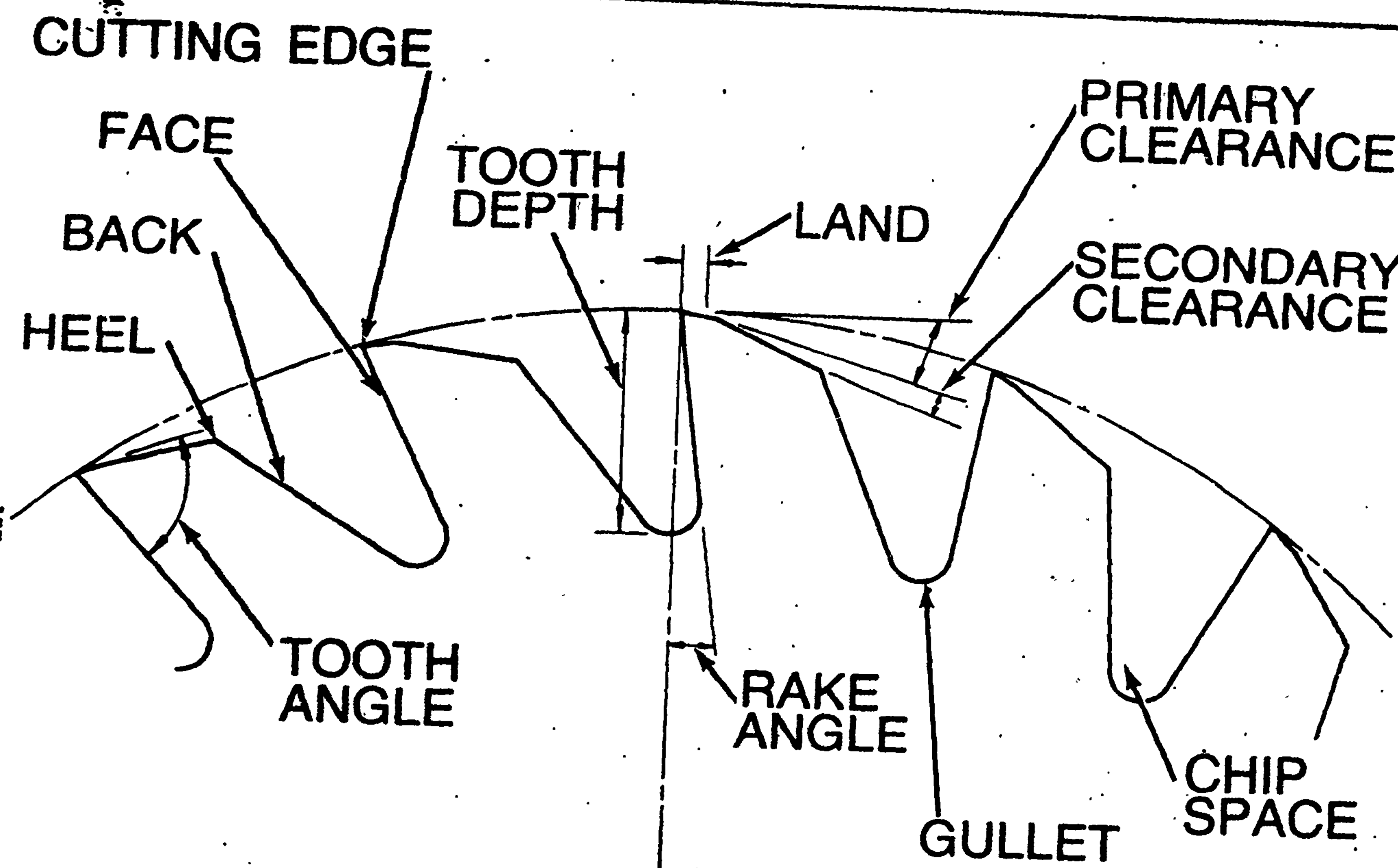
(a) CLIMB MILLING CUTTER



(b) CONVENTIONAL MILLING CUTTER

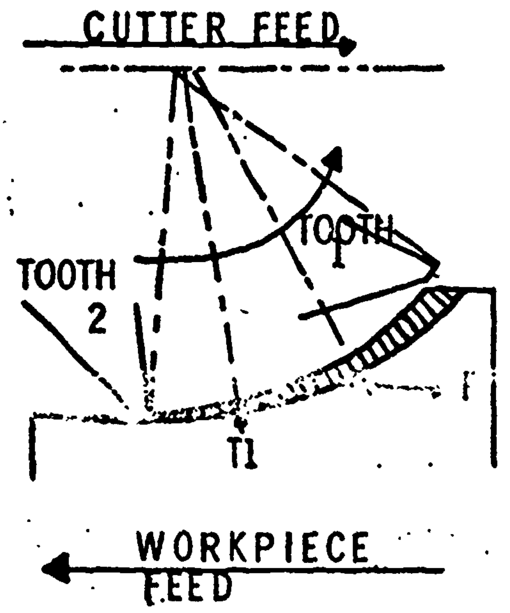
TYPES OF MILLING



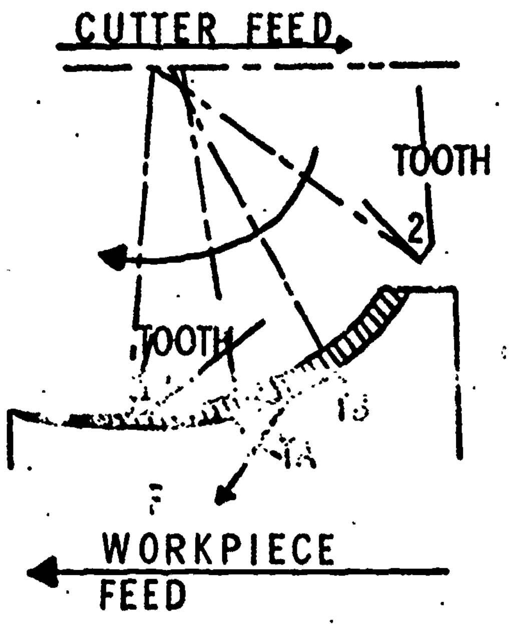


NOMENCLATURE OF MILLING CUTTER

CONVENTIONAL MILLING



CLIMB MILLING



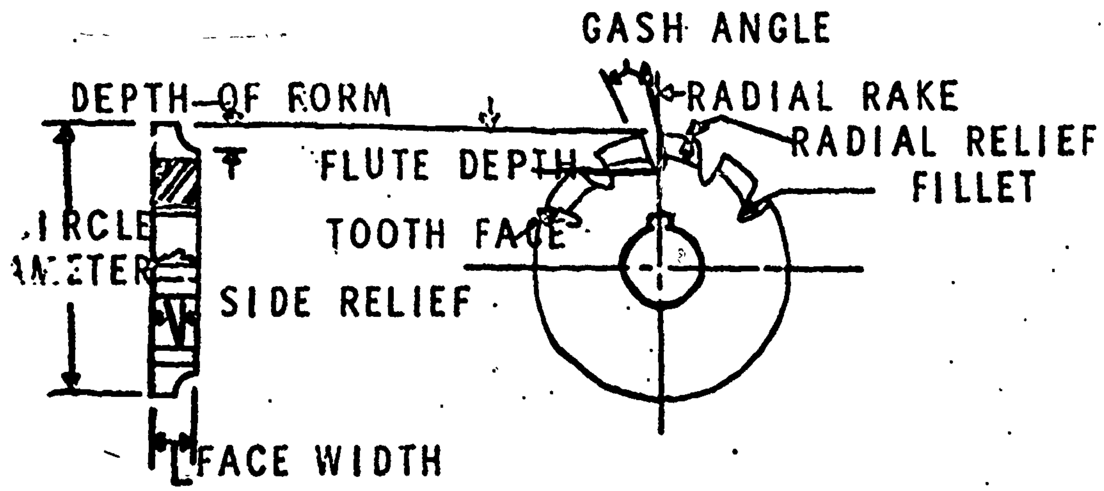
CLIMB MILLING OR DOWN MILLING *VS* CONVENTIONAL OR UP MILLING

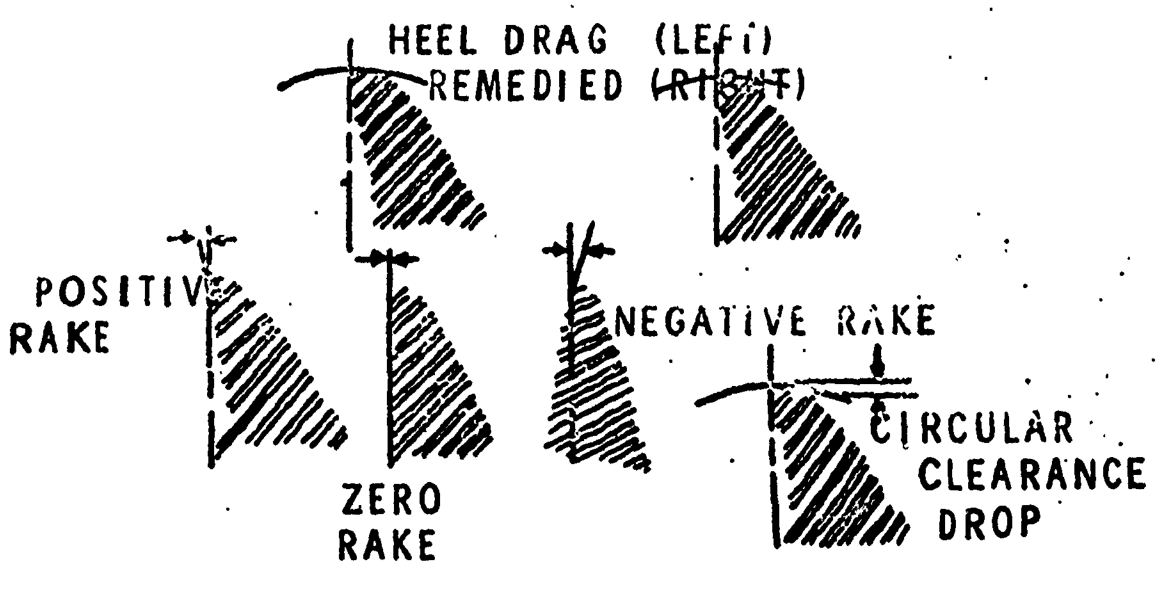
CLIMB MILLING WHEN APPLICABLE RESULTS IN BETTER FINISHES, FASTER FEEDS AND LONGER CUTTER LIFE. IT IS NOT APPLICABLE TO WORK HAVING A HARD SCALE TO CUT THROUGH AS A CAST OR SCALY FORGED SURFACE. ABRASION WOULD QUICK RUIN THE CUTTING EDGE. FASTER FEEDS AND BETTER FINISHES ARE POSSIBLE BECAUSE COMPONENT PARTS ARE UNDER A STEADY PRESSURE. IN CLIMB MILLING THE TOOTH HAS A QUICK AND POSTIVE ENTRANCE INTO THE METAL, NO RIDING OF THE PREVIOUSLY MILLED SURFACE AND HENCE LESS RUBBING. THIS PROMOTES LONGER CUTTER LIFE. CLIMB CUTTING CAN BE USED SUCCESSFULLY.ON ANY MACHINE IF ALL PARTS CAN BE KEPT TIGHT TO PREVENT BACKLASH OR PULLING IN. THIS IS ABSOLUTELY ESSENTIAL AND CAN BE DONE ON MACHINES OF THE OLDER TYPE IF CAREFUL.

CONVENTIONAL CUTTING HAS A TENDENCY TO LIFT THE WORK FROM ITS BEARING, THUS REDUCING RIGIDITY. AS THE CUTTER REVOLVES AGAINST THE WORK IT IS FORCED AWAY BY THE FEED OF THE WORK WHICH, IN TURN, SPRINGS THE ARBOR. THIS ACTION CONTINUES UNTIL THE RESISTANCE OF THE ARBOR OVERCOMES THE RESISTANCE OF THE WORK TO CUTTING, AND THE TOOTH ENTERS THE WORK. THIS RUBBING ACTION AND THE HEAT GENERATED BY IT PROBABLY BREAKS THE CUTTER DOWN QUICKER THAN THE ACTUAL CUTTING. CONVENTIONAL CUTTING SHOULD BE USED ON MATERIALS HAVING A HARD OR SCALY SURFACE, AS THIS IS THE LESSER OF TWO EVILS. CLIMB CUTTING IS MORE DAMAGING THAN THE ABRASIVE ACTION OF CONVENTIONAL MILLING. ABARSION DUE TO SCALE IN CONVENTIONAL MILLING IS MINIMIZED BY THE FACT THAT THE SCALE IS LIFTED OFF AHEAD OF THE CUTTING EDGE.

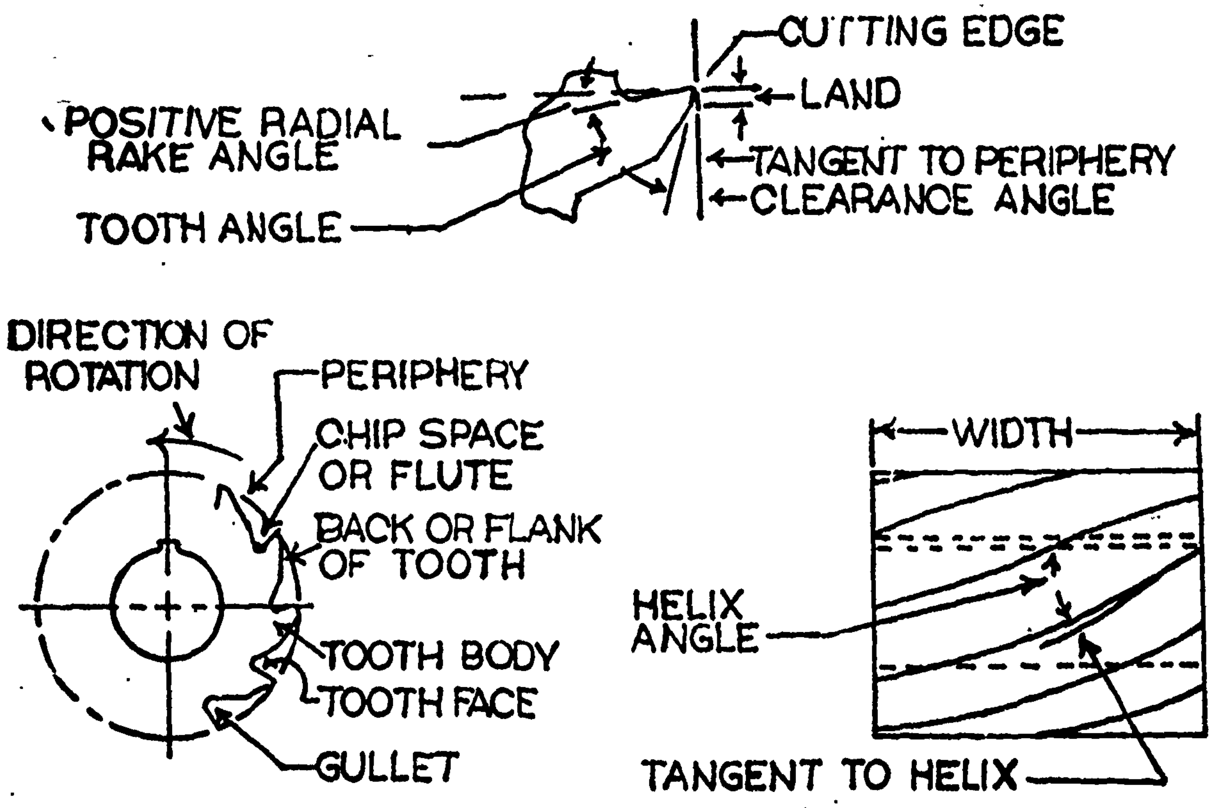
172

NOMENCLATURE OF FORM MILL CUTTERS

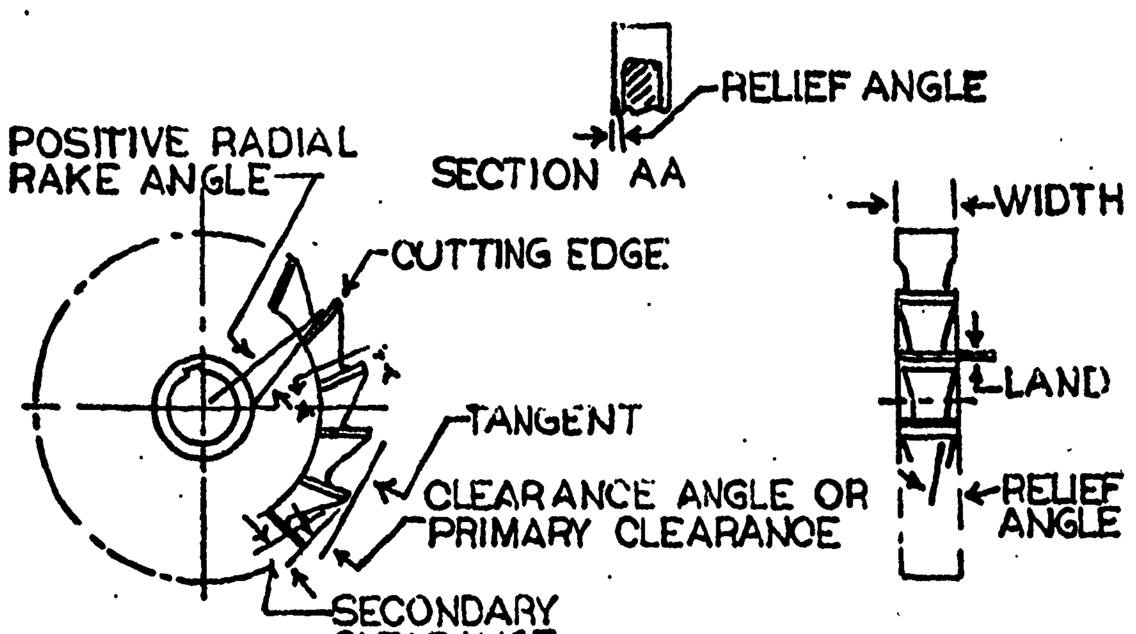




MULTI POINT CUTTING TOOLS



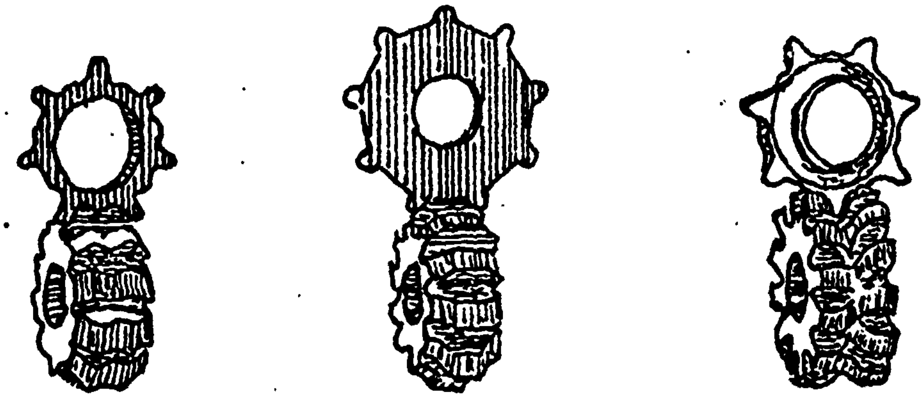
PLAIN MILLING CUTTER WITH NOMENCLATURE



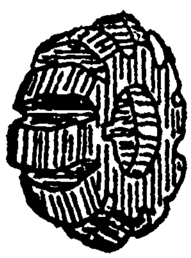
SIDE MILLING CUTTER WITH NOMENCLATURE

M-5 50

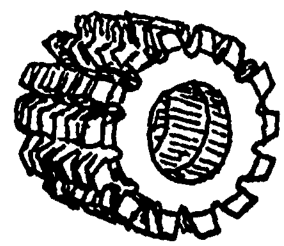
MULTI POINT FORM CUTTERS



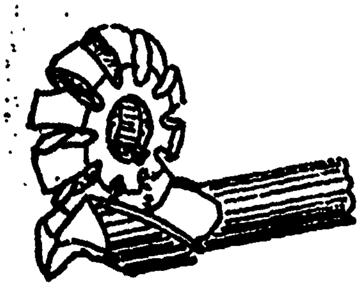
SPROCKET CUTTERS



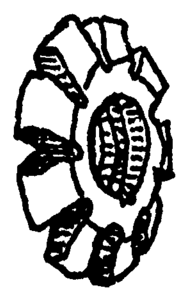
TAP & REAMER CUTTER



WORM GEAR HOB



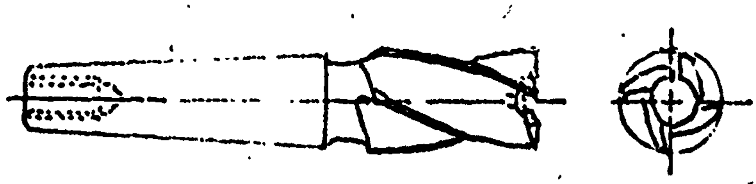
TWIST DRILL CUTTER



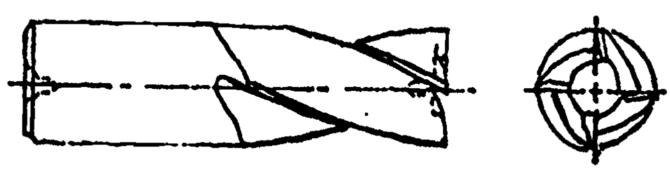
GEAR TOOTH CUTTER

M-5-51

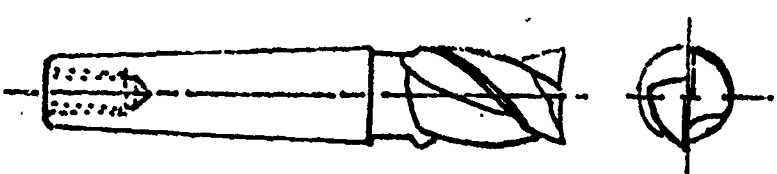
STANDARD MILLING CUTTERS



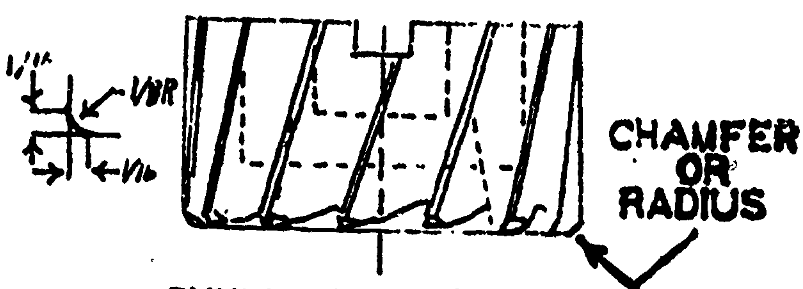
FOUR FLUTE TAPER SHANK END MILL



FOUR FLUTE STRAIGHT SHANK END MILL



TWO FLUTE TAPER SHANK END MILL

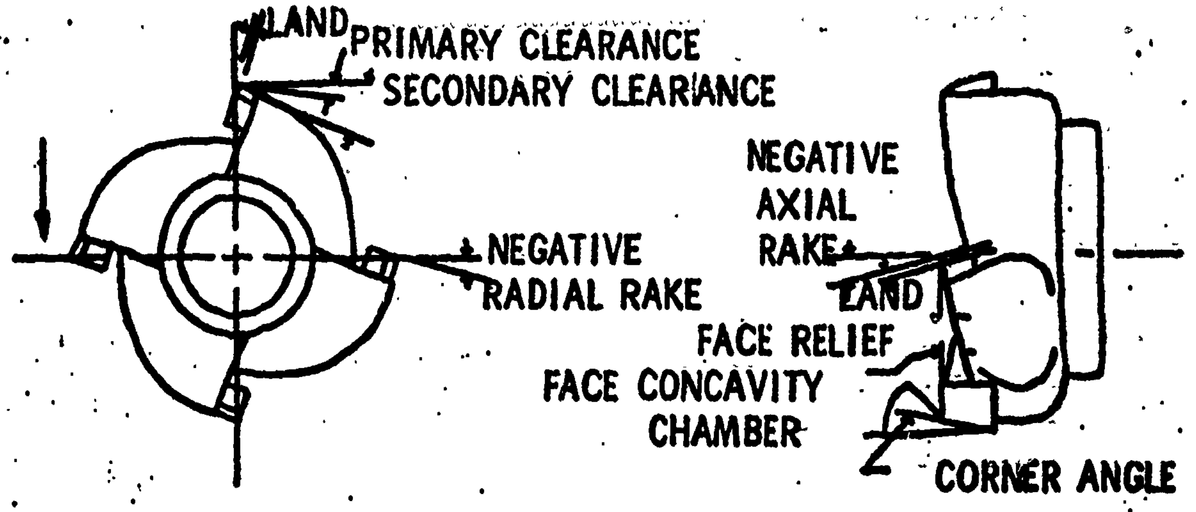


SHELL END MILL

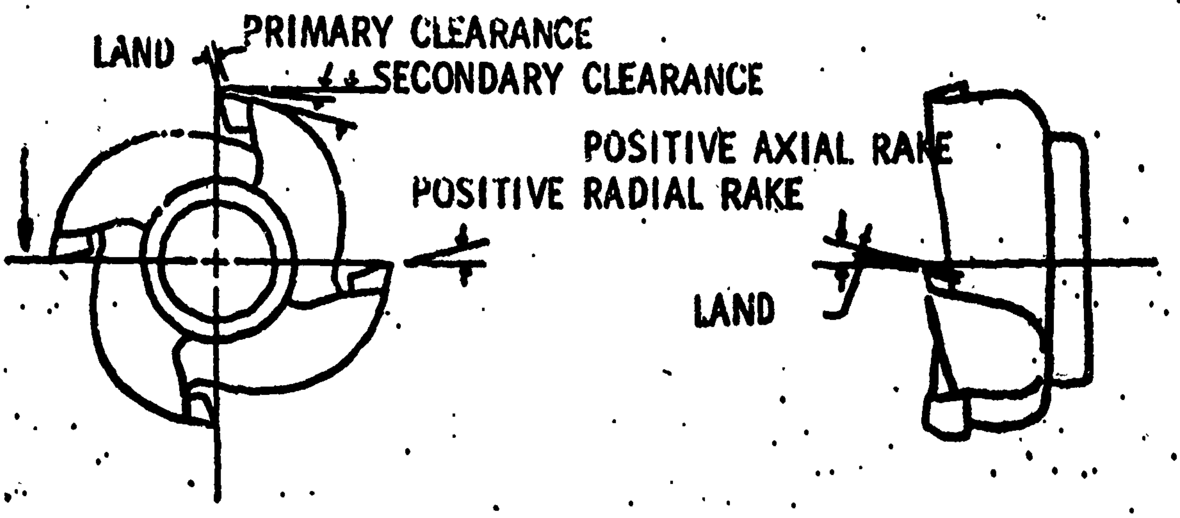
M-5

MILLING CUTTER NOMENCLATURE

NEGATIVE ANGLE CUTTER R.H.

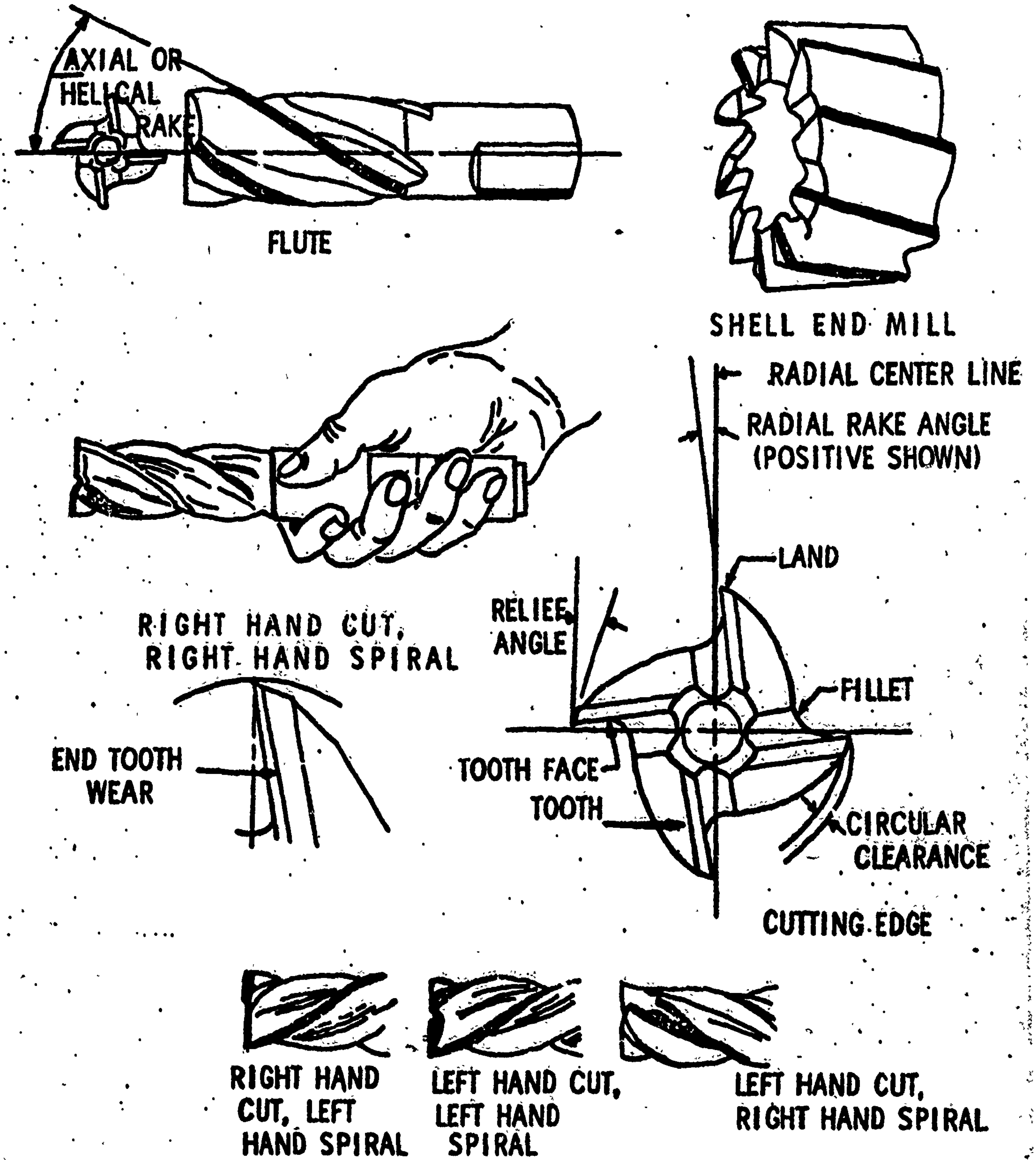


POSITIVE ANGLE CUTTER R.H.

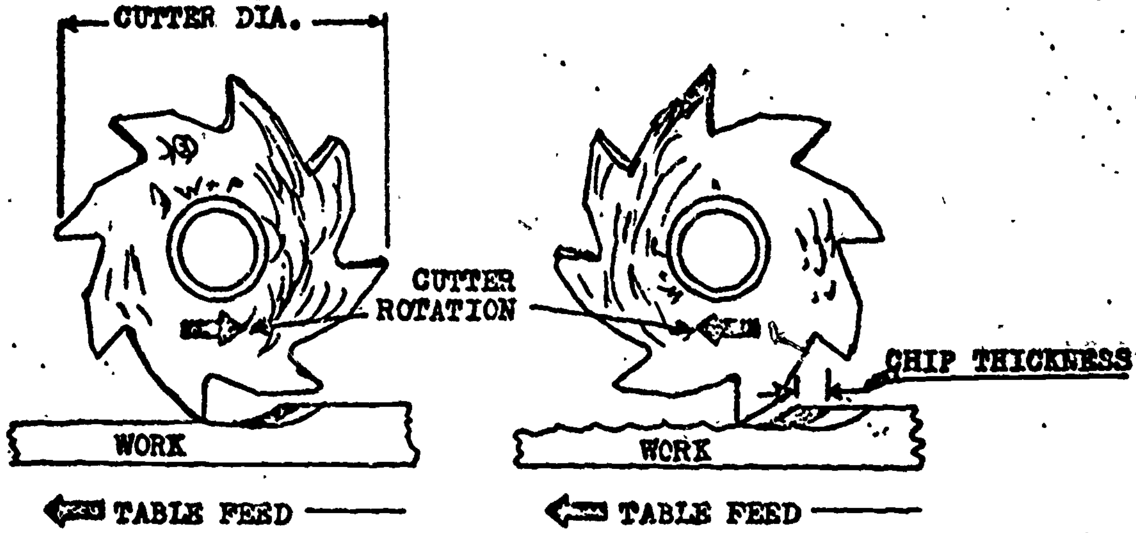


M-U 115

END MILL NOMENCLATURE



M-10-118



CONVENTIONAL OR UP

DOWN OR CLIMB

SPEED: R.P.M. OF CUTTER, ALSO DEFINED AS SURFACE FEET PER MINUTE MEASURED ON THE CUTTER PERIPHERY.

FORMULA: R.P.M. $= \frac{4 \times S}{d}$ WHERE RPM - REVOLUTIONS PER MINUTE

4 CONSTANT USED INSTEAD OF Pi (π)

S SURFACE SPEED OF CUTTER

d DIAMETER OF CUTTER

EXAMPLE: FIND THE RPM OF AN 4" dia. CUTTER, MACHINING A MATERIAL WITH A 250 ft/min. CUTTING SPEED.

SOLUTION: RPM $= \frac{4 \times 250}{4} = 250$ ∴ RPM is 250

FEED: THE RATE AT WHICH THE WORK IS MOVED AGAINST THE ROTATING CUTTER, USUALLY EXPRESSED IN INCHES PER MINUTE.

FORMULA: IMP = No. TEETH ON CUTTER X CHIP THICKNESS X R.P.M.

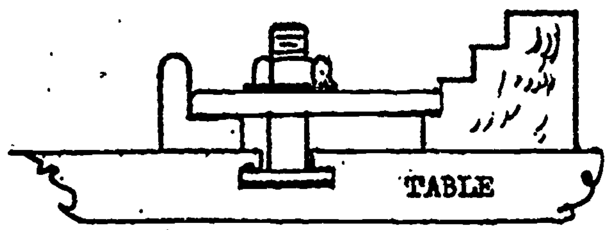
EXAMPLE: WHAT WILL BE THE FEED FOR A CUTTER WITH 24 TEETH, A CHIP THICKNESS OF .003", ROTATING 90 RPM.

SOLUTION: IMP = 24 X .003 X 90 = 6.480 OR 6 3/8 INCHES PER MIN.

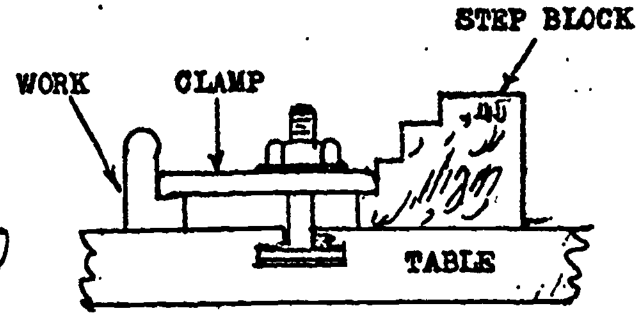
CAUTIONS: CUTTER MATERIAL, MATERIAL TO BE MACHINED, TYPE OF SET UP, AND CONDITION OF MACHINE GOVERN USE OF THESE FORMULAS.

CORRECT AND INCORRECT METHODS OF

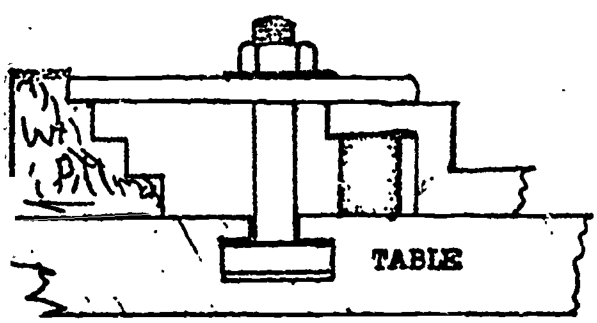
CLAMPING WORK TO TABLE



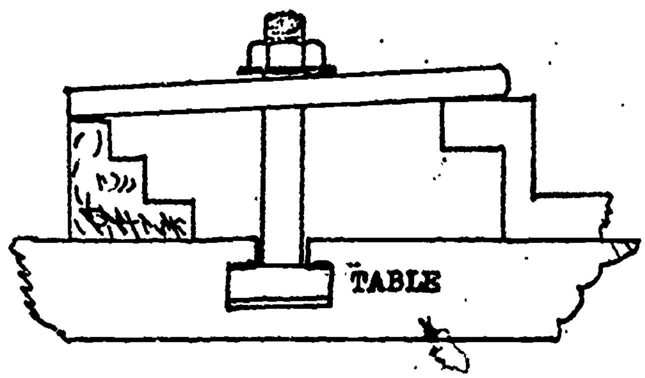
CORRECT



INCORRECT



CORRECT



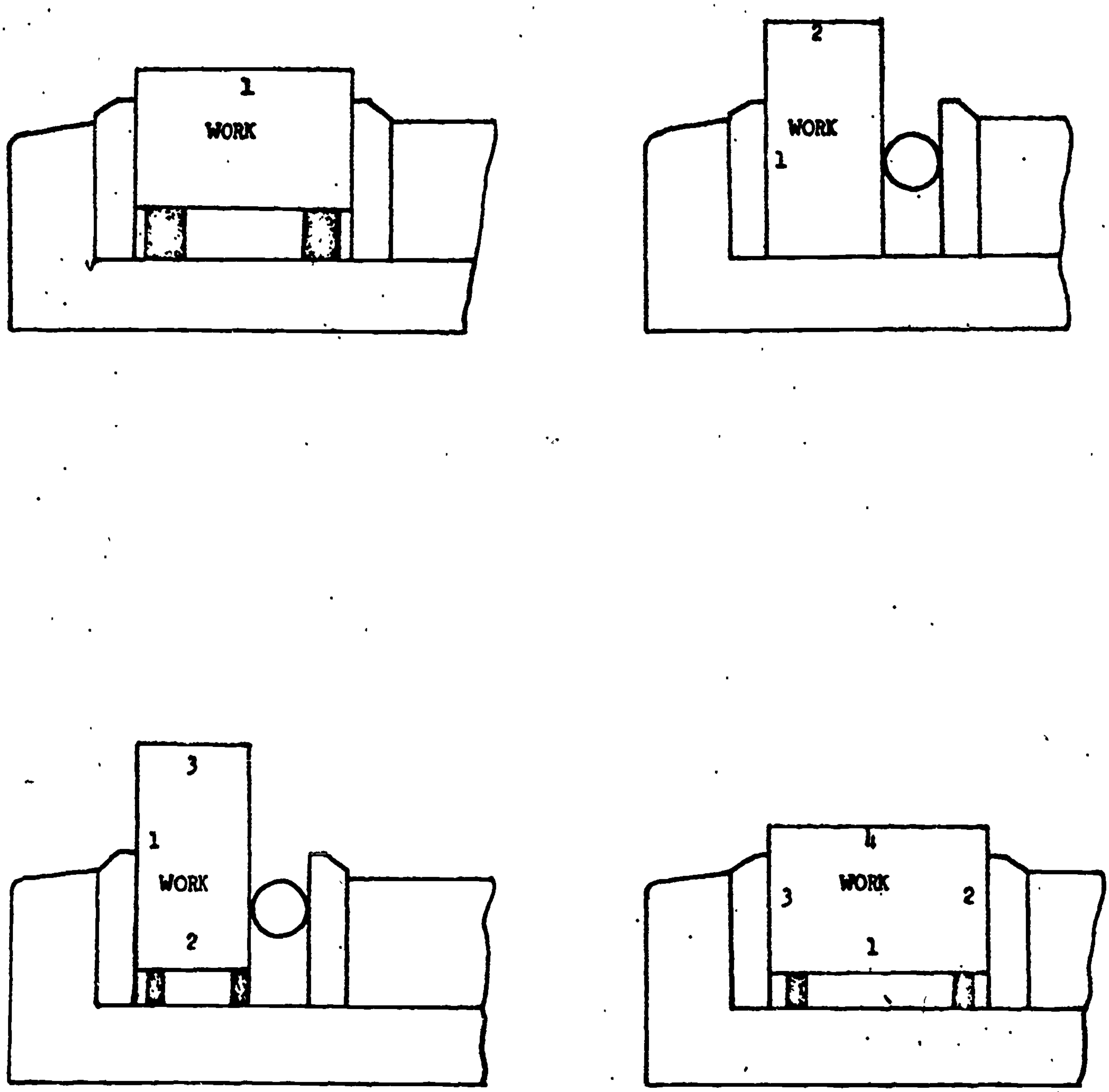
INCORRECT

RULE: ALWAYS HAVE CLAMP AS NEAR PARALLEL AS POSSIBLE WITH TABLE SURFACE AND THE CLAMPING BOLT AS CLOSE TO THE WORK AS IS PRACTICAL.

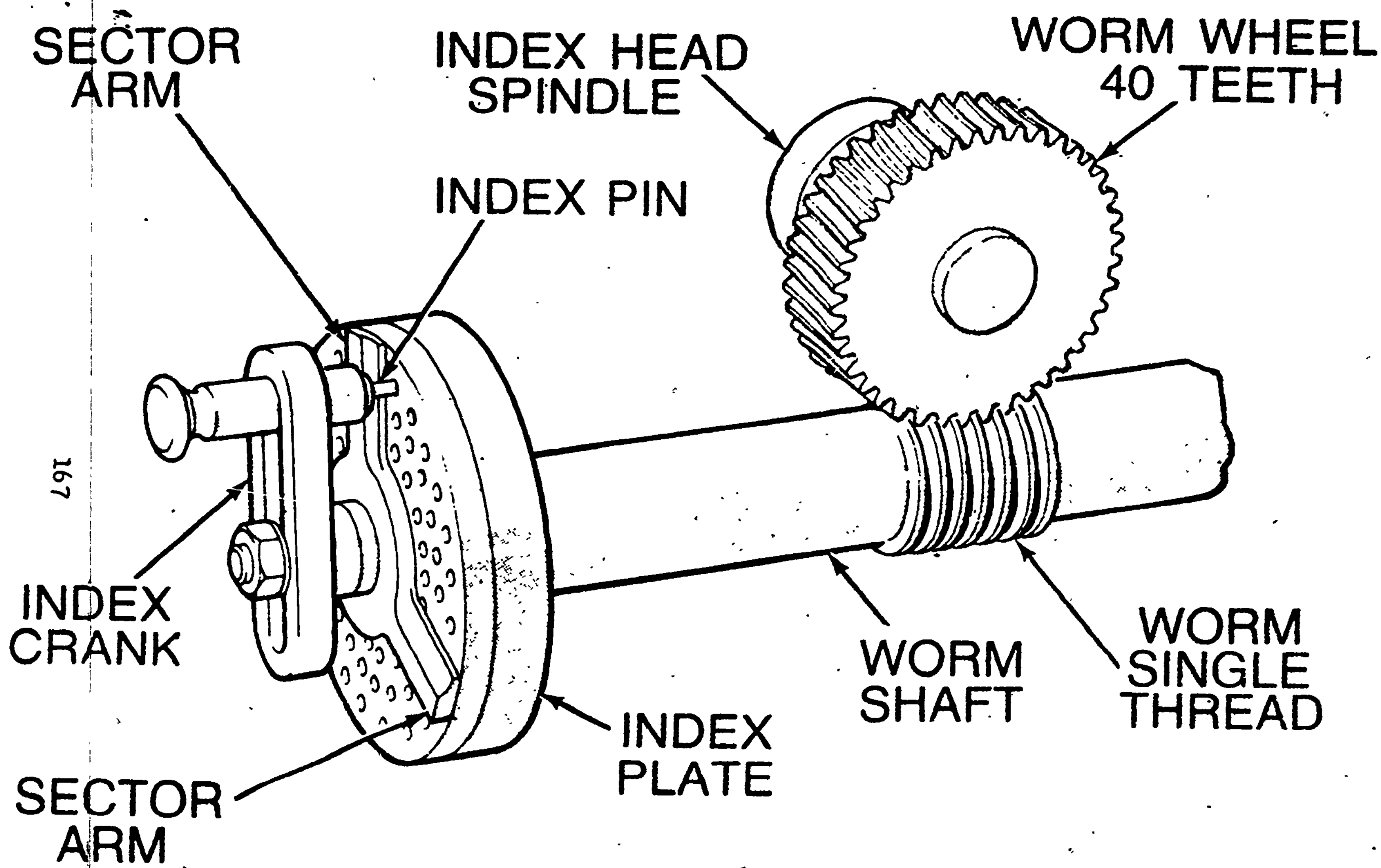
IN THIS MANNER MORE PRESSURE WILL BE EXERTED ON THE WORK THAN ON THE STEP BLOCK THEREBY CREATING MORE HOLDING POWER.

OTHER CLAMPING DEVICES: V - BLOCKS, ANGLE PLATES, CHUCK, SCREW HEEL CLAMP, STRAP CLAMP, GOOSENECK CLAMP, SCREW JACK, U-CLAMP, SINGLE FINGER CLAMP, C-CLAMP, AND FIXTURES.

172



SETUP FOR MACHINING A BLOCK SQUARE AND PARALLEL



INTERNAL MECHANISM OF AN INDEX HEAD

ASSIGNMENT SHEET

TITLE: TO FIND THE R.P.M. AND CUTTING SPEED OF A MILLING MACHINE

UNIT: MILLING MACHINE

OCCUPATION: MACHINIST

OBJECTIVE: To give the student practice in solving problems used in caculating the cutting speed or the R.P.M.'s of a milling machine.

REFERENCES: 1. Anderson-Tatro Shop Theory. New York: McGraw-Hill Book Co, Inc.
2. Axelrod, Aaren-Machine Shop Mathematics New York: McGraw-Hill Book Co., Incl

DIRECTIONS: Read the above references, study the following examples and work the problems below.

Cutting speed of a milling cutter is the number of feet traveled by a point on the revolving cutter in one minute, or the speed in feet per minute of a point on its outer surface. It is the product of the circumference of the cutter times the revolution per minute.

Formula:

$$RPM = \frac{4 \times CS}{D}$$

$$CS = \frac{D \times R.P.M.}{4}$$

Examples: To find the cutting speed of a $2\frac{1}{2}$" milling cutter revolving at the rate of 60 RPM

$$CS = \frac{D x RPM}{4} = \frac{2\frac{1}{2} x 60}{4} = 37\tfrac{1}{2} \text{ ft per min.}$$

To find what RPM is required for a 6" cutter at 80 feet per minute?

$$RPM = \frac{4 x CS}{D} = \frac{4 x 80}{6} = 63\ 1/3 RPM$$

PROBLEMS:

1. Determine the proper cutting speed for a $4\frac{1}{2}$" cutter at 100 feet per minute.

2. Give the RPM at which a 3" mill should run to have a speed of 150 feet per minute.

3. A ½" end mill at 80 feet per minute calls for a RPM of what?

4. What RPM is required for a 1" end mill at 100 feet per minute cutting speed?

5. The cutting speed of a 1" end mill cutting mild steel at 80 feet per minute should be what?

ASSIGNMENT SHEET

TITLE: SHAPES AND SHAPER OPERATIONS

UNIT: SHAPER WORK

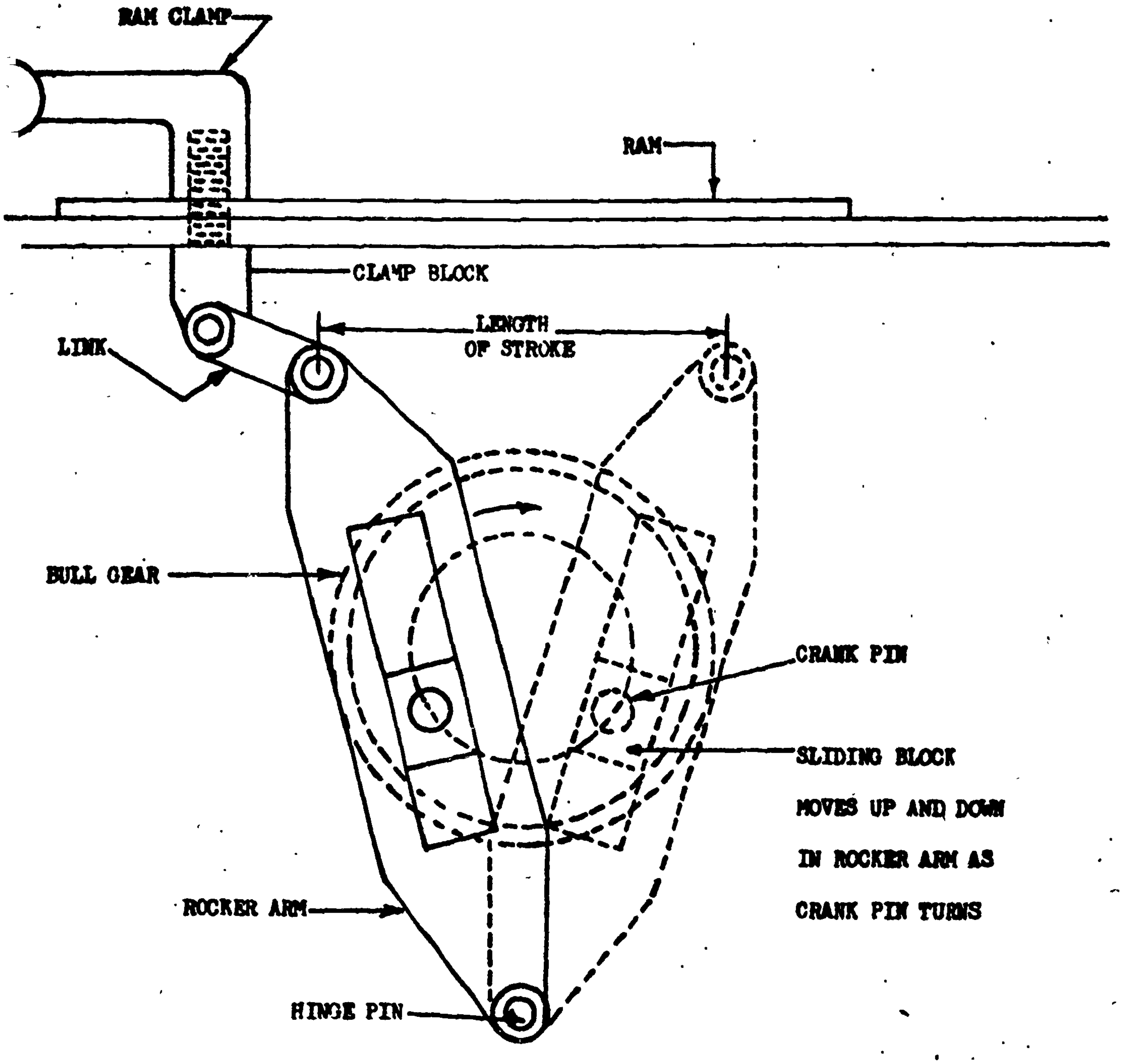
OCCUPATION: MACHINIST

OBJECTIVE: To acquaint the student with shapers and shaper operations.

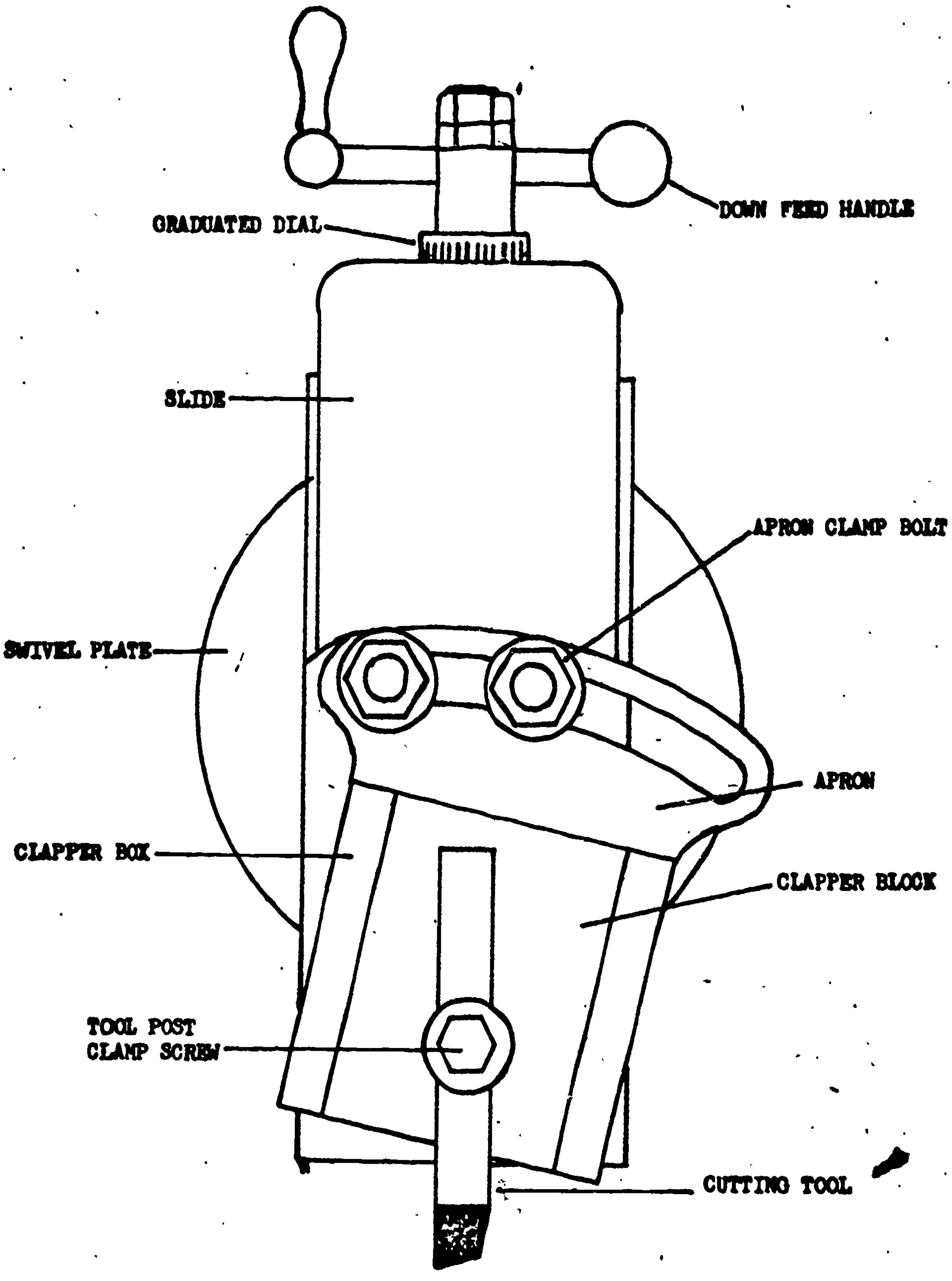
REFERENCE: Anderson-Tatro. Shop Theory. McGraw-Hill Book Co., Inc. Chapter 11, pages 279-301.

QUESTIONS:

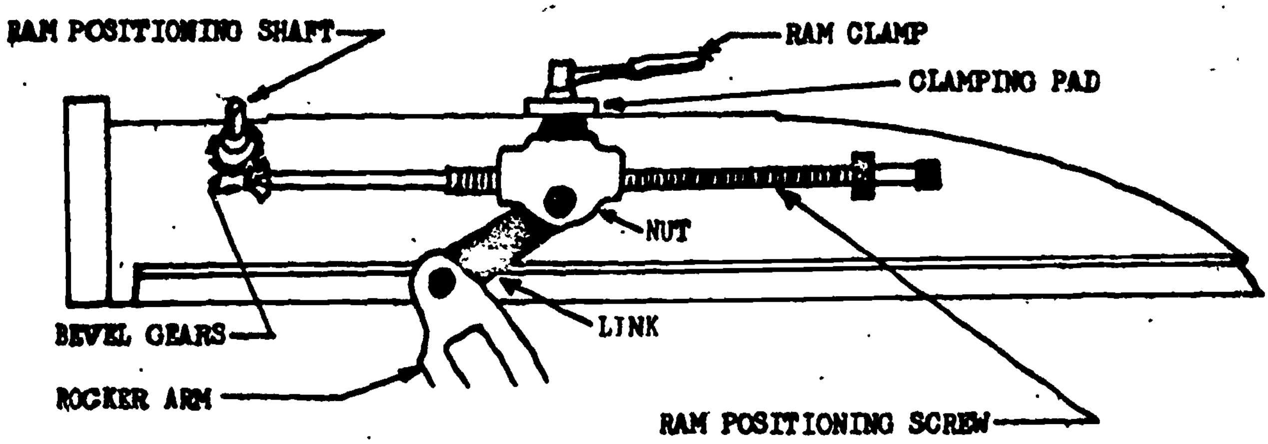
1. Explain the function of the following parts of a shaper.
   a. Base
   b. Frame
   c. Ram
   d. Tool head
   e. Table
2. How is the size of a shaper determined?
3. How does a universal shaper differ from a regular shaper?
4. What table movements are found on vertical shapers?
5. How is the length of stroke of a shaper determined?
6. How is feed expressed on a shaper?
7. What is the purpose of the clapper box?
8. List five operations that can be done on a shaper?
9. How is cutting speed expressed on a shaper?
10. Discuss the different methods of holding materials while shaping.



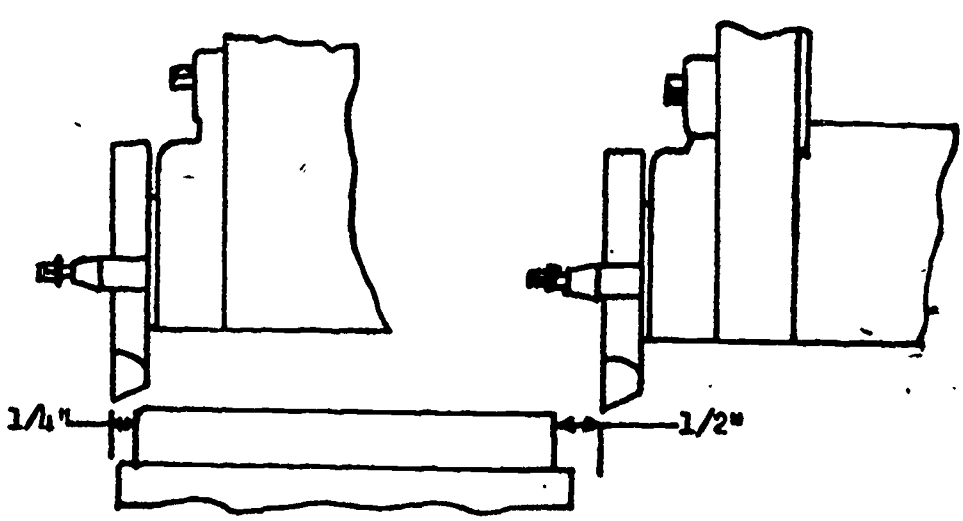
DRIVE OF A CRANK SHAPER



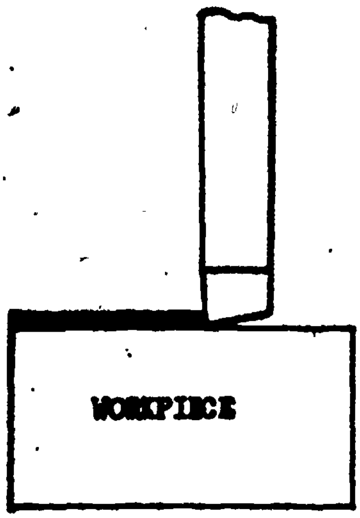
PARTS OF A SHAPER TOOLHEAD



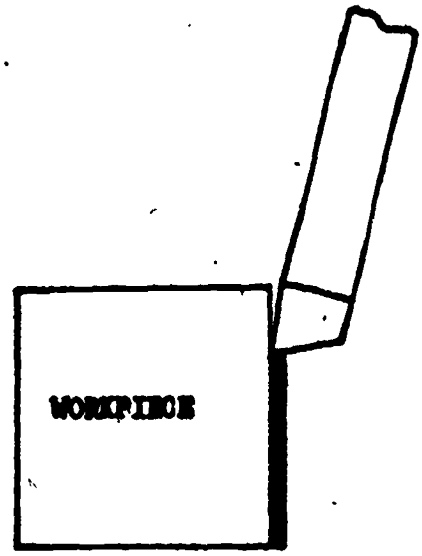
RAM POSITIONING CONTROLS



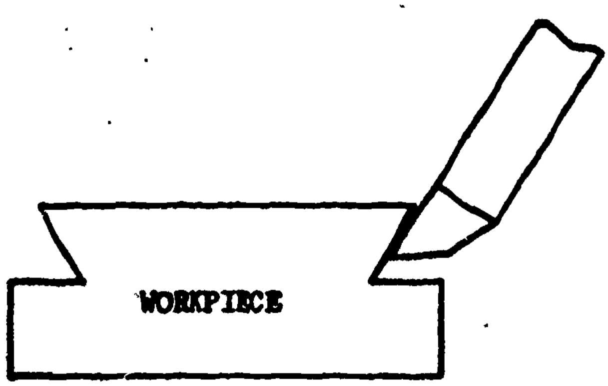
POSITION OF RAM IN RELATION TO LENGTH OF WORKPIECE



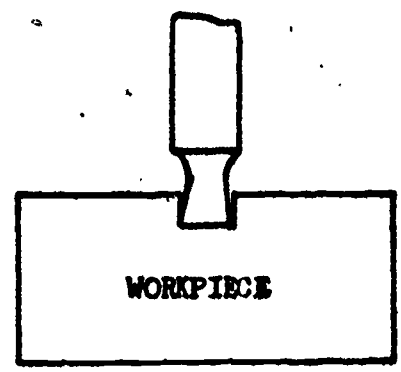
HORIZONTAL CUT



VERTICAL CUT

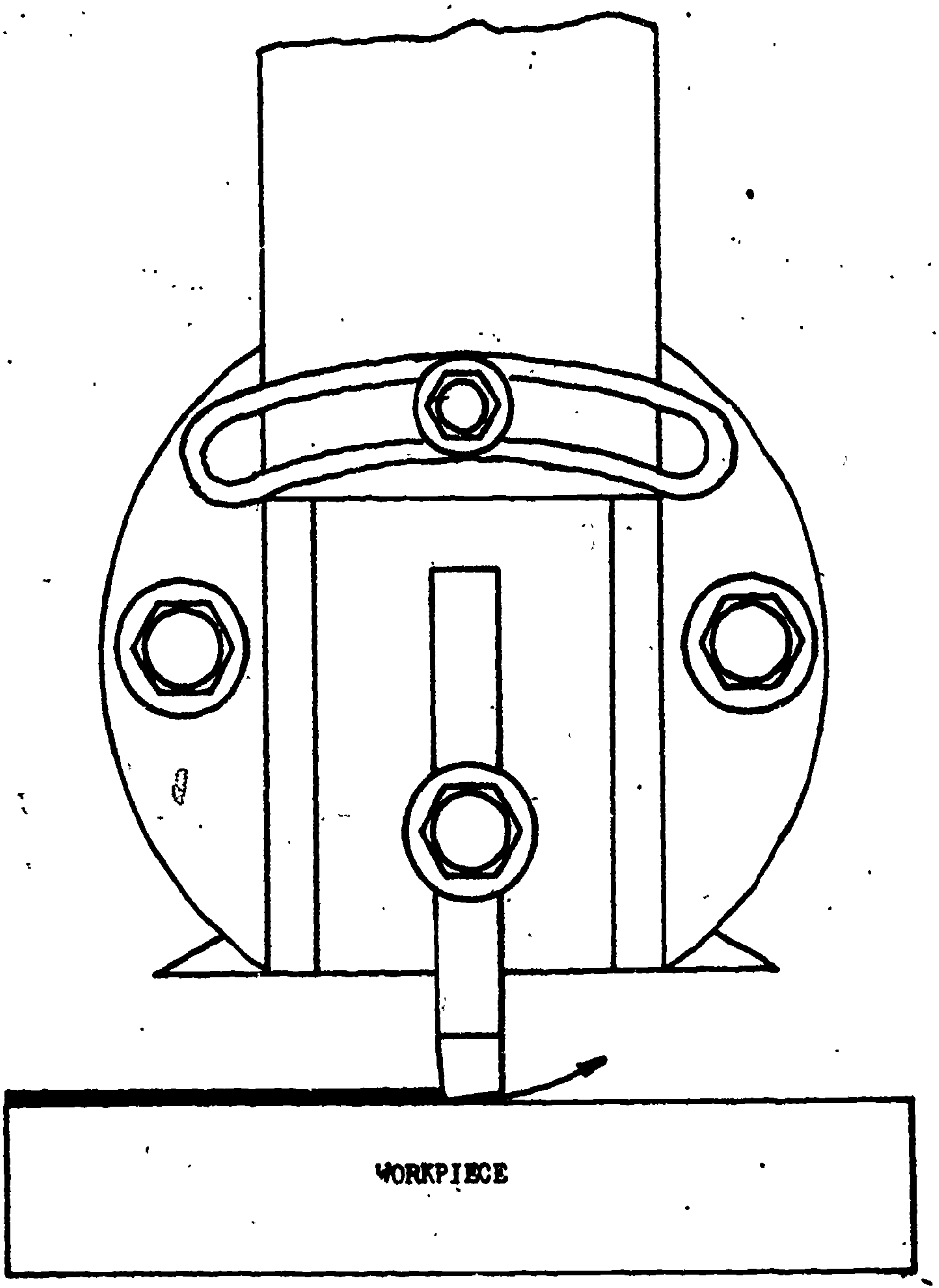


ANGULAR CUT



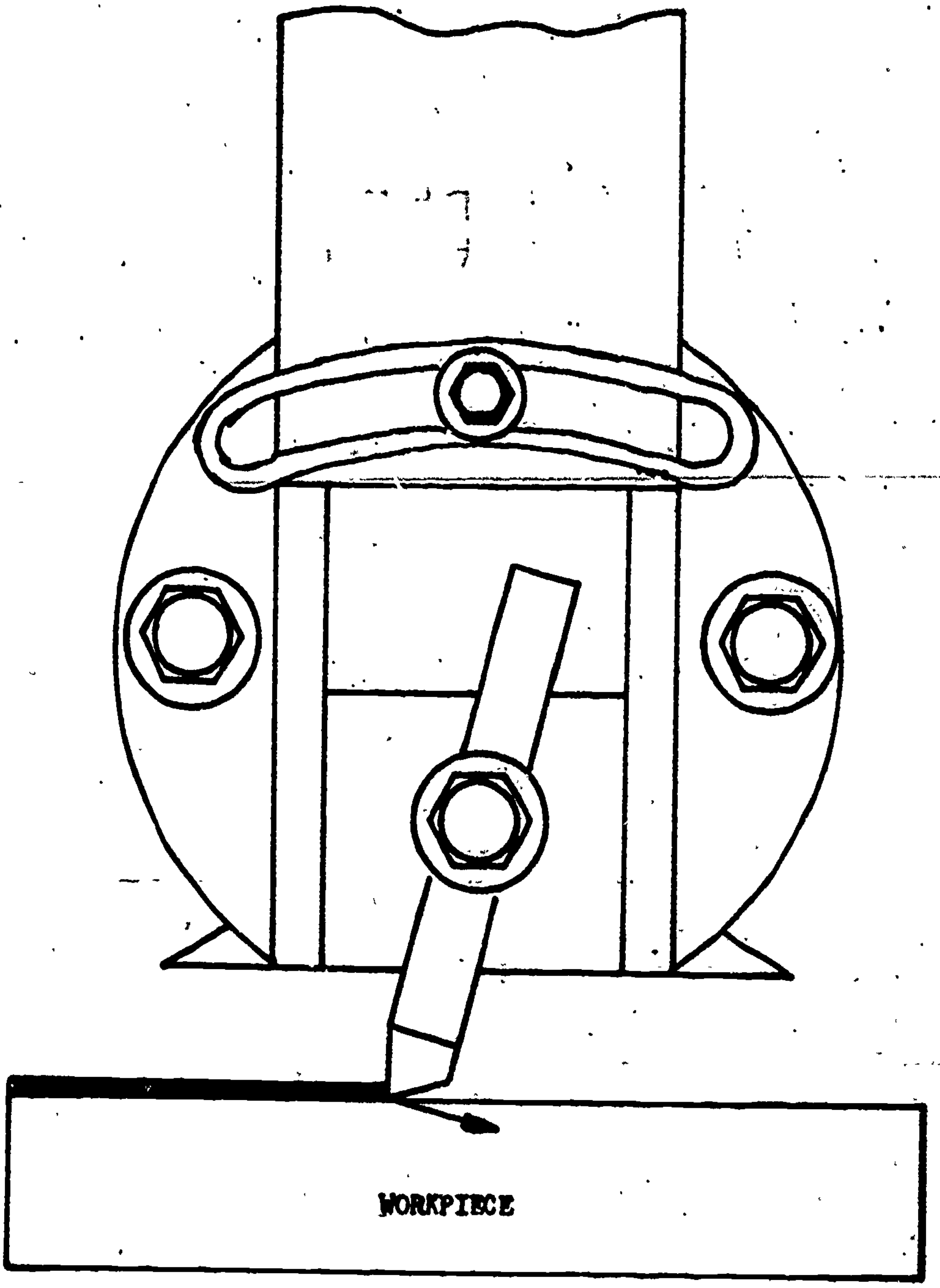
SLOTTING OPERATION

SOME COMMON SHAPER OPERATIONS



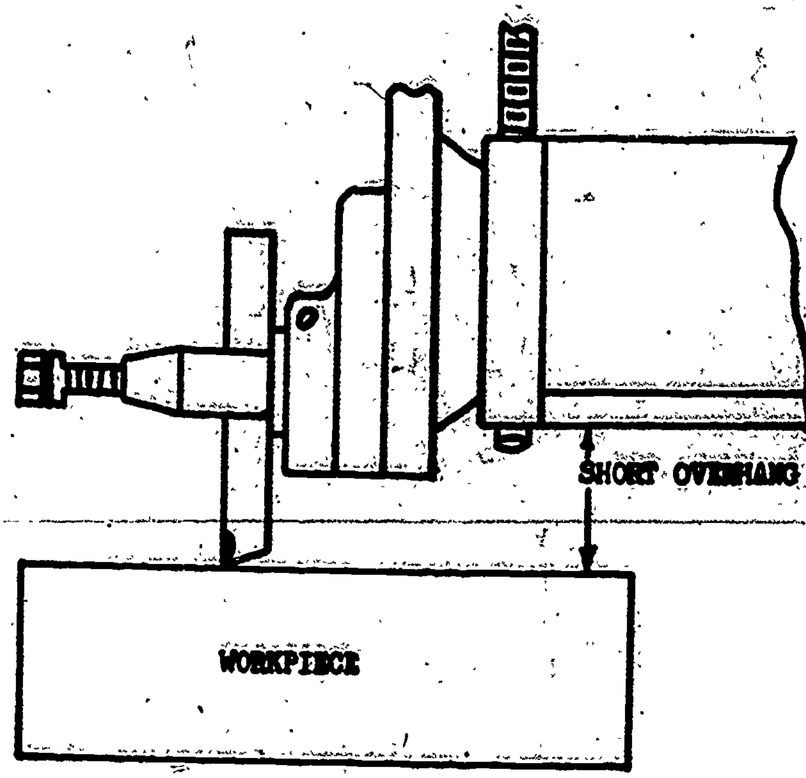
CORRECT WAY TO MOUNT TOOL

TOOL WILL SWING AWAY FROM WORK IF IT SLIPS UNDER CUTTING PRESSURE



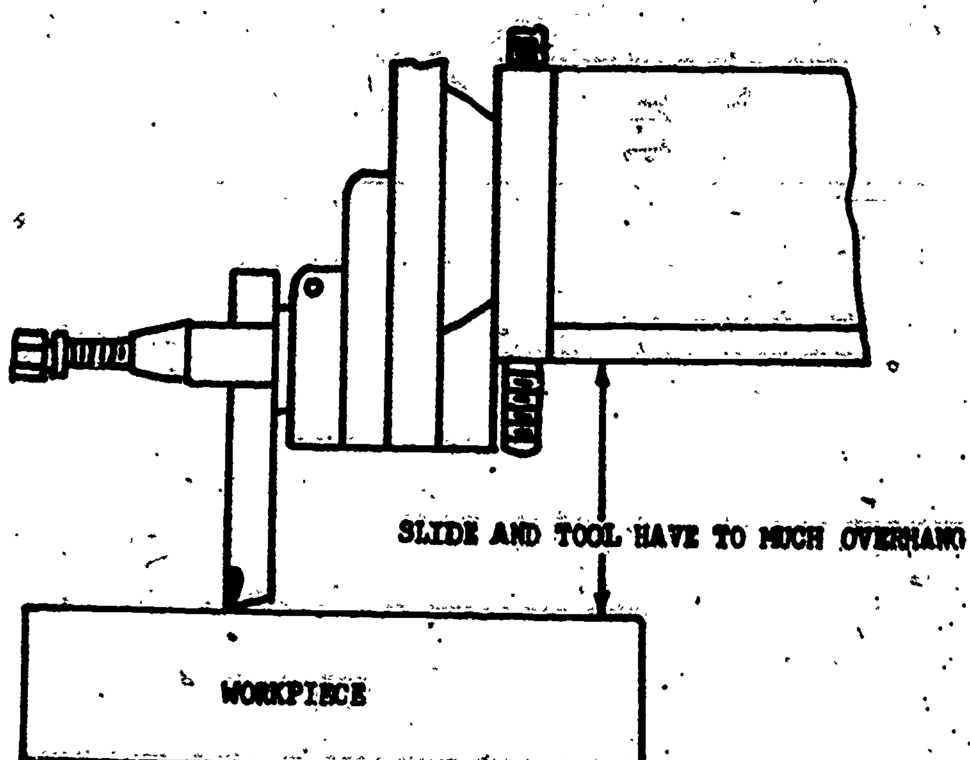
INCORRECT WAY TO MOUNT TOOL

THE TOOL WILL DIG INTO THE WORKPIECE IF IT SLIPS UNDER CUTTING PRESSURE



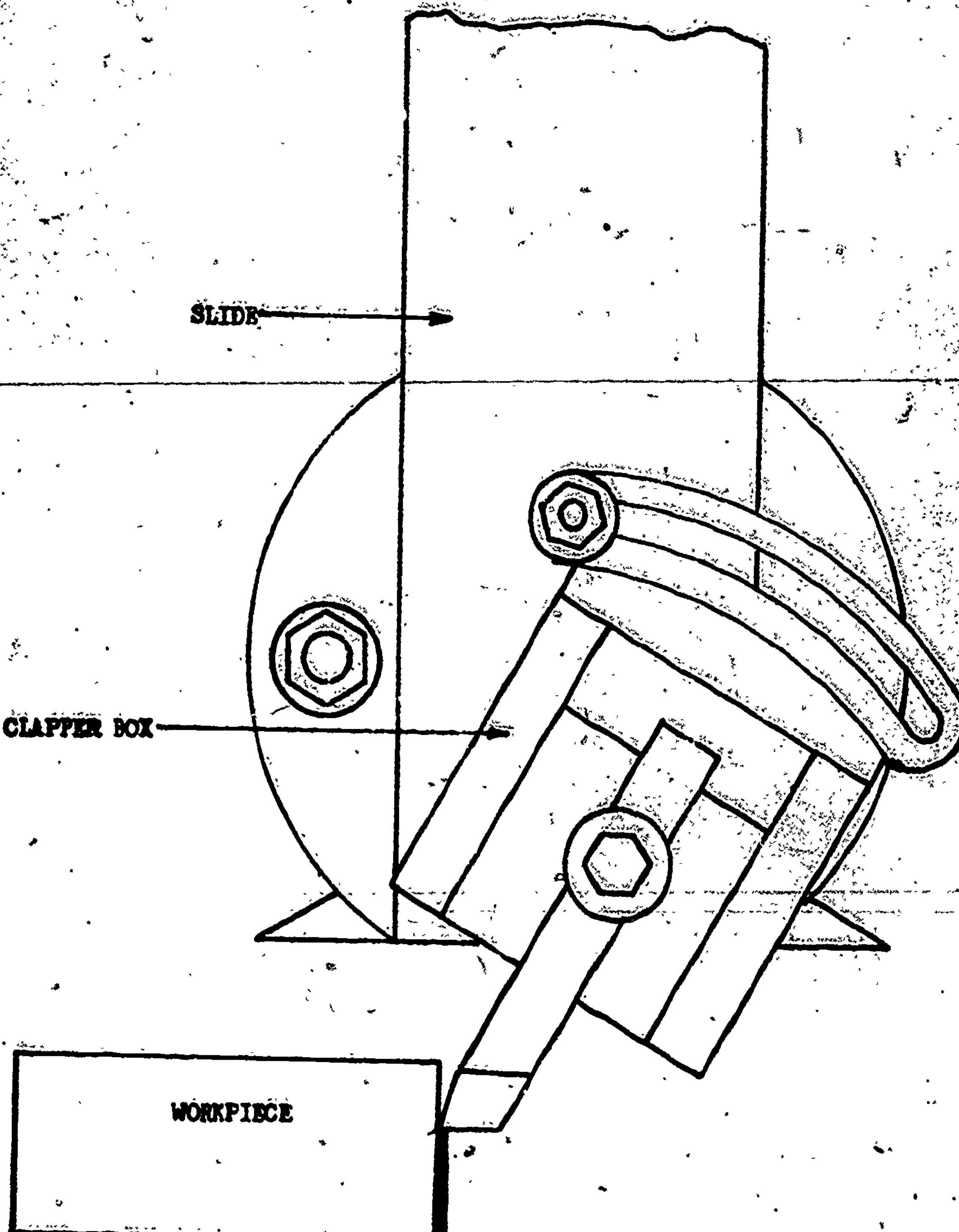
RIGHT

FOR MAXIMUM RIGIDITY, KEEP THE SLIDE UP AND GRIP THE TOOL SHORT

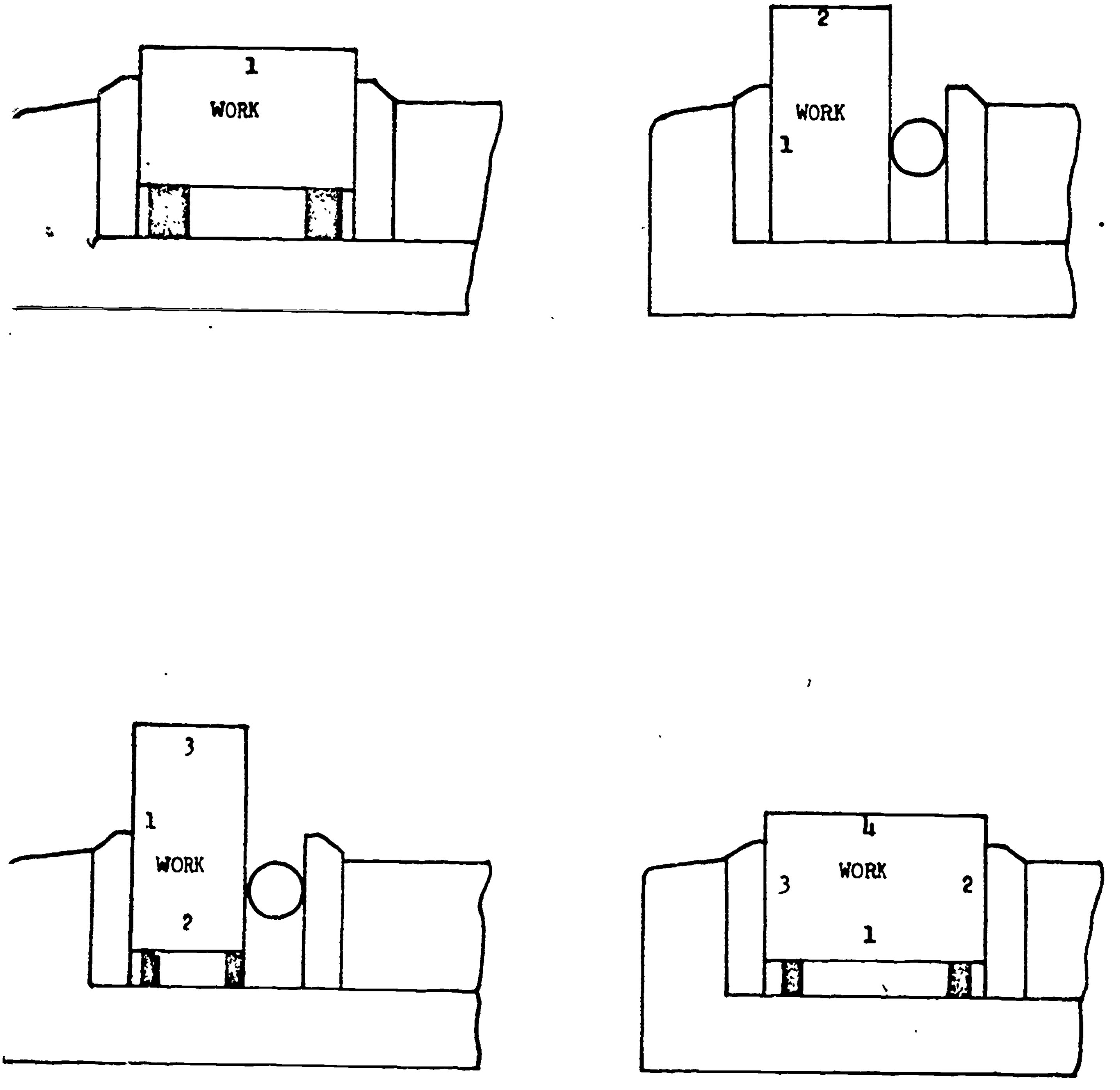


WRONG

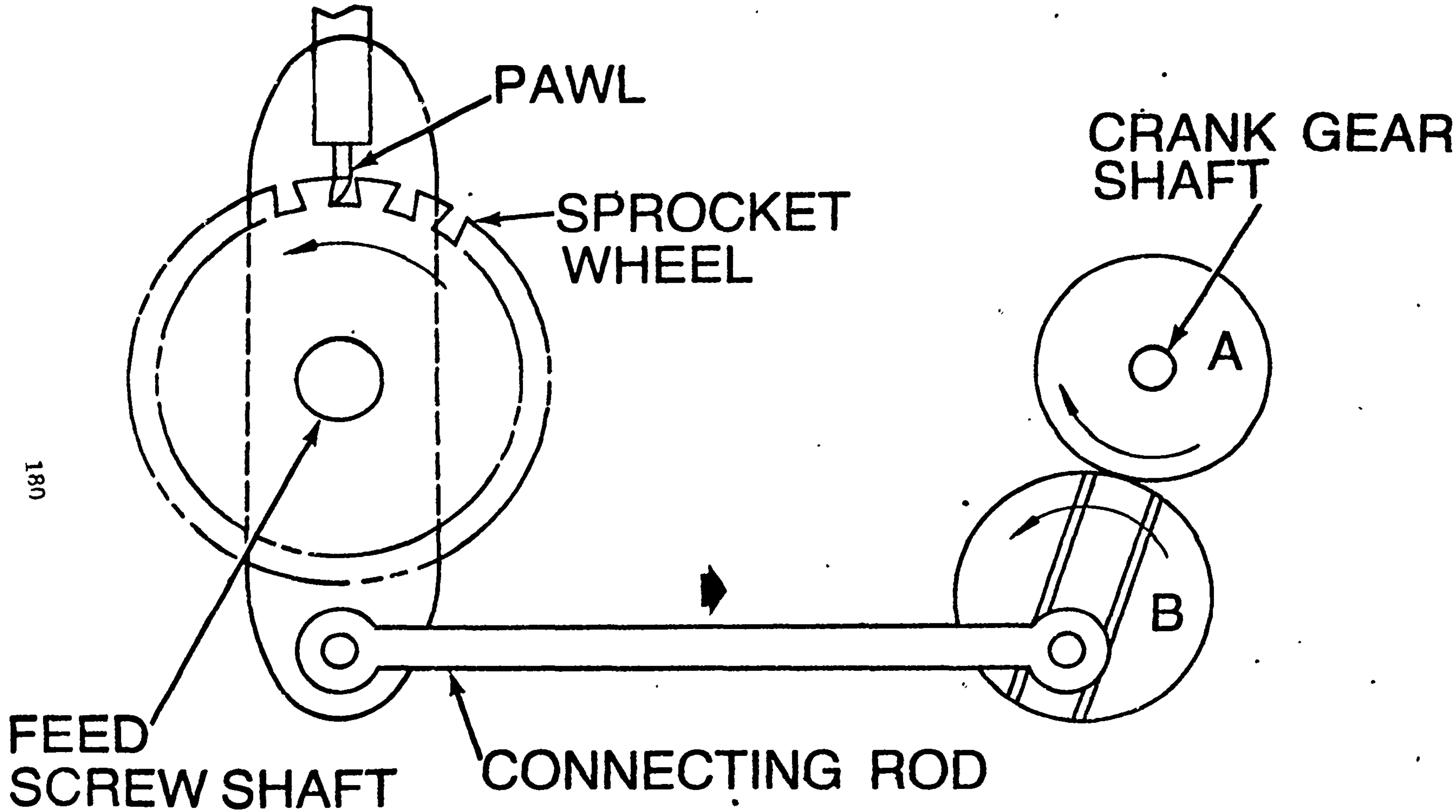
TO MUCH OVERHANG OF THE TOOL AND SLIDE MAY CAUSE THE TOOL TO SPRING AND CHATTER



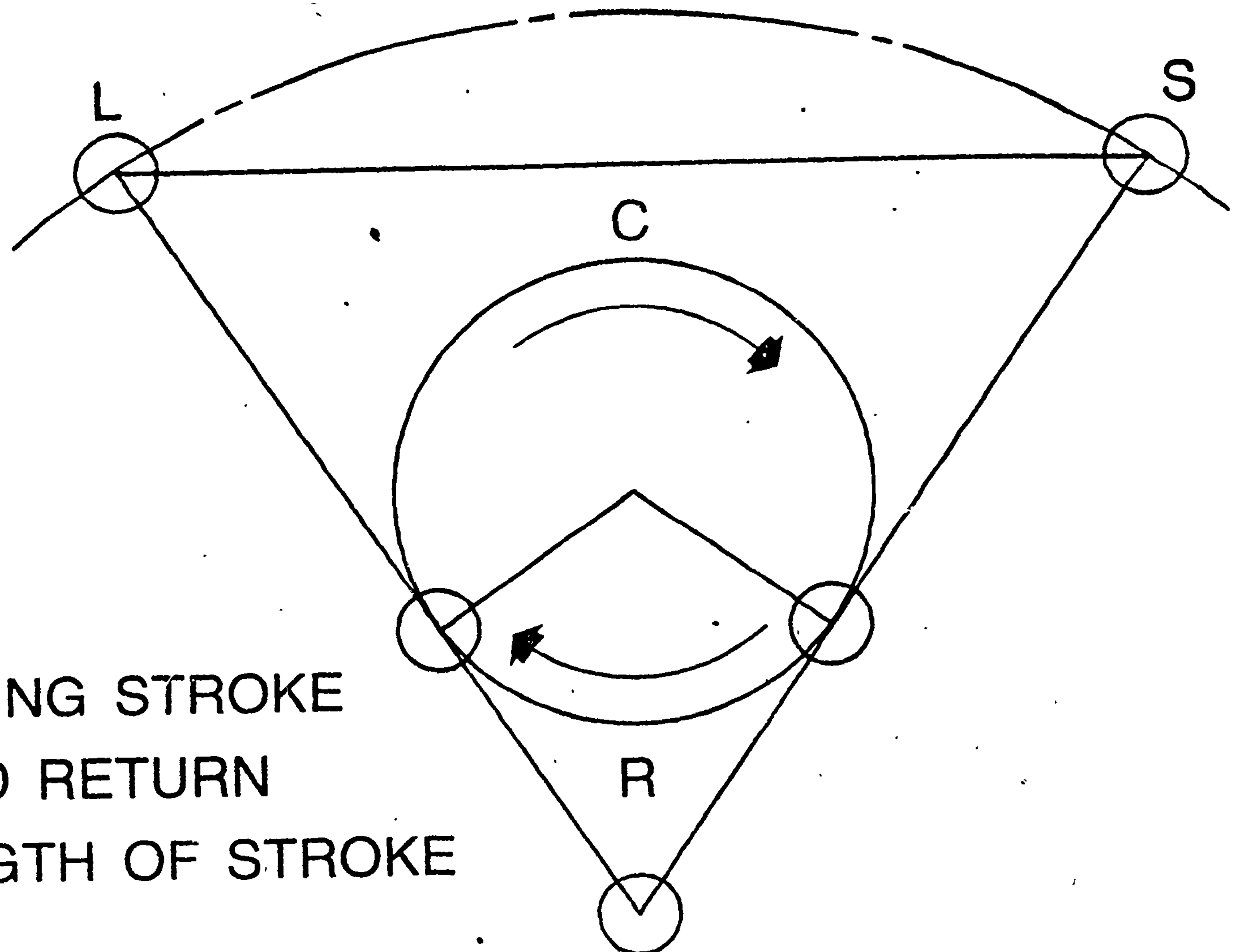
POSITION OF SLIDE AND CLAPPER BOX FOR MAKING A VERTICAL CUT



SETUP FOR MACHINING A BLOCK SQUARE AND PARALLEL



SHAPER CROSSFEED MECHANISM



C = CUTTING STROKE

R = RAPID RETURN

LS = LENGTH OF STROKE

PRINCIPLE OF THE QUICK RETURN MECHANISM ON A SHAPER

ASSIGNMENT SHEET

TITLE: PLANER AND PLANER OPERATIONS

UNIT: PLANER WORK

OCCUPATION: MACHINIST

OBJECTIVE: To acquaint the student with planer and planer operations.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 12.

DIRECTIONS: Read the above reference and answer the following questions.

1. Explain the function of the following parts of a planer.

    a. Bed
    b. Table
    c. Housing
    d. Crossrail
    e. Saddle
    f. Toolhead
    g. Gang

2. How is the size of a planer determined?

3. How does a double housing type planer differ from the open side type planer?

4. What type of cutting tools are used on the planer?

5. How does the planer tool head compare with the shaper tool head?

6. How is work held on a planer?

7. What is a planer gage and how is it used?

8. What accessories are available for planer work?

9. How does a gong tool differ from a regular tool?

10. How is the speed and feed of a planer determined?

ASSIGNMENT SHEET

TITLE: TYPES OF GEARS

UNIT: GEARS

OCCUPATION: MACHINIST

OBJECTIVE: To familiarize the student with the different types of gears, their uses and how they are made.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 13, pages 313-328.

DIRECTIONS: Read the above reference, answer the following questions and define the following terms.

QUESTIONS:

1. What is a spur gear?
2. How are spur gears used?
3. What is the pressure angle of spur gears teeth?
4. What is the purpose of a gear rack?
5. How is the chordal thickness of a gear tooth measured?
6. How many spur gear cutters are there in a set?
7. What is a bevel gear?
8. How are bevel gears used?
9. What are miten gears?
10. What is a helical gear?
11. What are the advantages of helical gears?
12. What are herringbone gears?
13. What is worm gearing?
14. Explain how and where worm gears are used.
15. How is the ratio of worm gears calculated?

16. List the methods used to make gears.

17. What is the corrected addendum of a gear tooth?

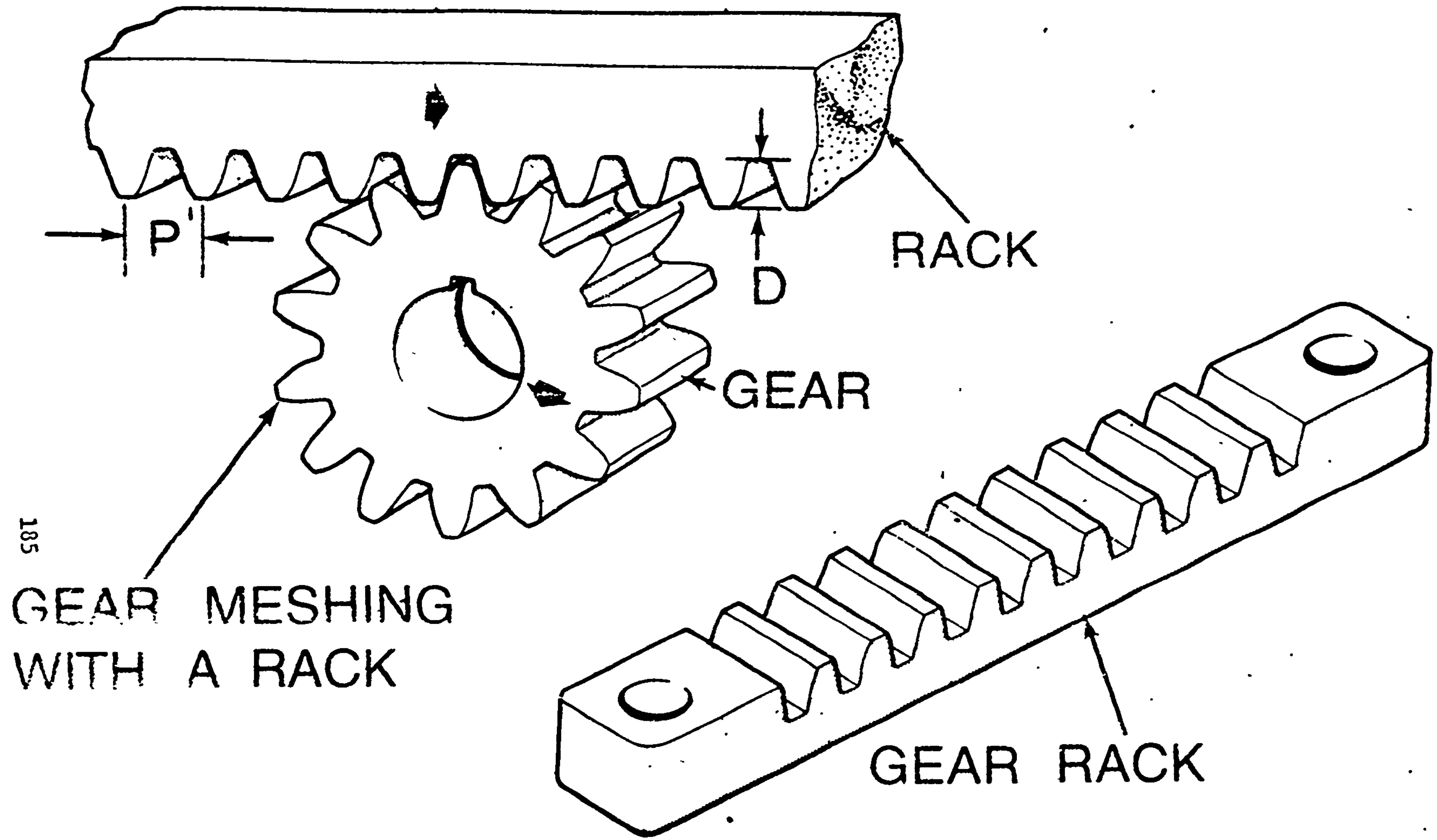
18. What is the ratio of a plain index head?

19. How can gears that cannot be index on a plain index head be indexed?

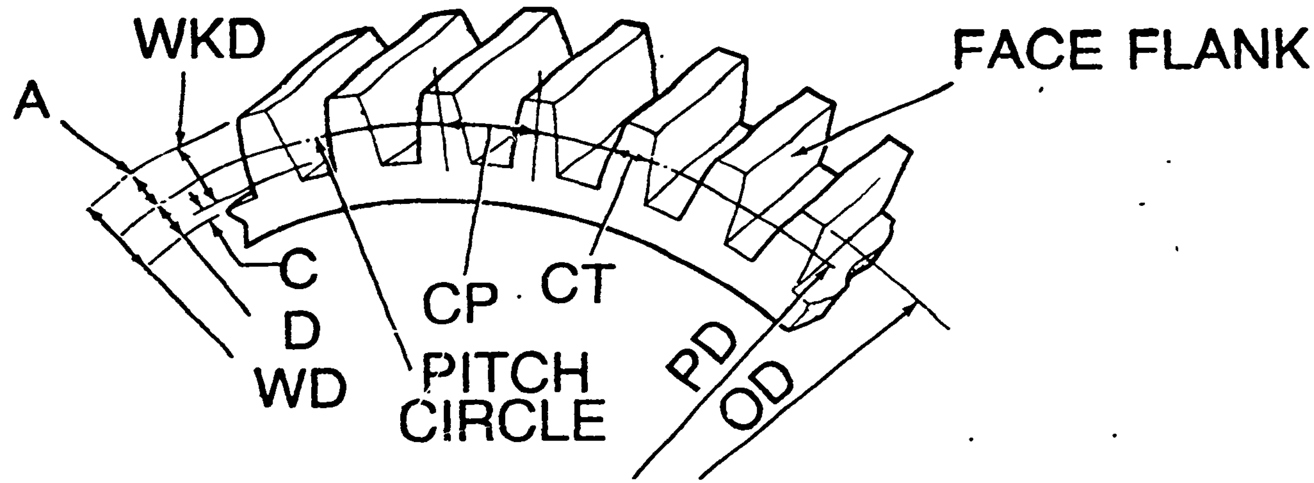
20. What is defentrial indexing?

TERMS:

1. Pitch circle
2. Pitch diameter
3. Diametral pitch
4. Circular pitch
5. Addendum
6. Dedendum
7. Clearance
8. Gear-tooth vernier
9. Gear sector
10. Hole
11. Pinion
12. Rack
13. Driven gear
14. Driving gear
15. Indexing

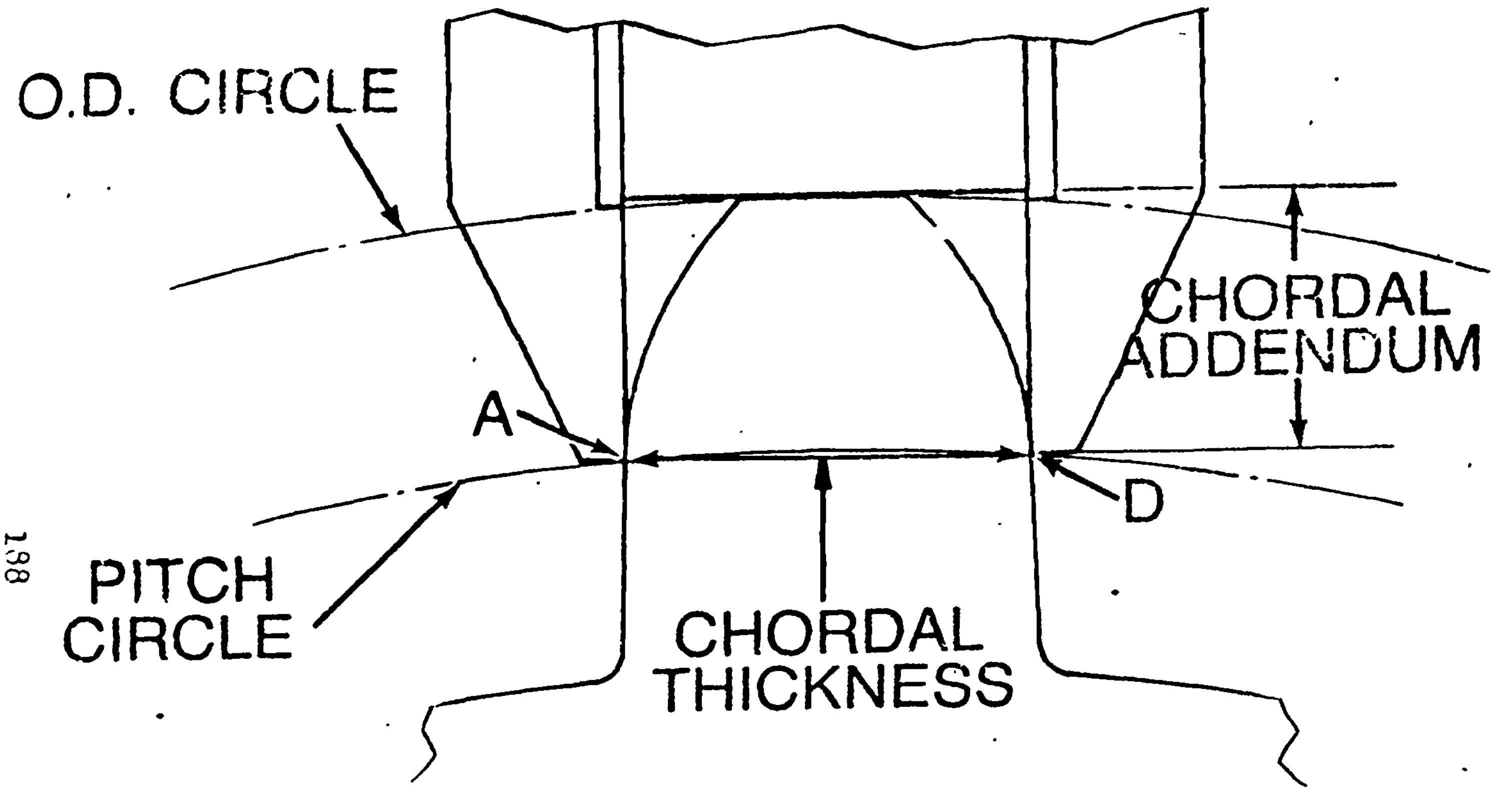


SPUR GEAR AND RACK



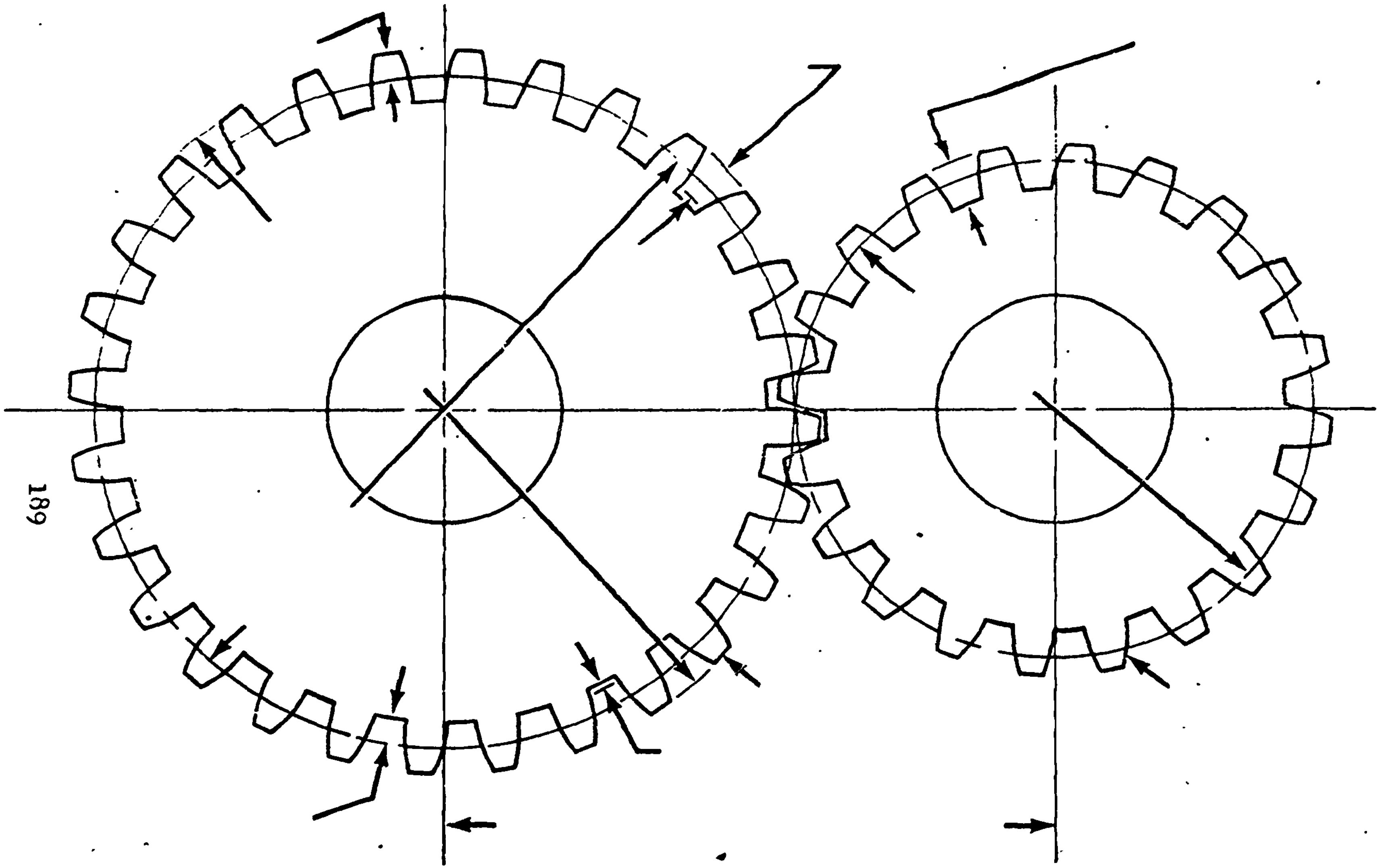
SPUR GEAR NOMENCLATURE AND FORMULAE

PITCH DIAMETER	$PD = \frac{N}{DP}$	ADDENDUM	$A = \frac{1}{DP}$	ROOT DIAMETER	$RD = PD - 2D$
NUMBER OF TEETH	$N = PD \times DP$	DEDENDUM	$D = \frac{1.157}{DP}$	WHOLE DEPTH	$WD = \frac{2.157}{DP}$
DIAMETRAL PITCH	$DP = \frac{N}{PD}$	CLEAR	$C = \frac{.157}{DP}$	WORKING DEPTH	$WKD = \frac{2}{DP}$
CIRCULAR PITCH	$CP = \frac{\pi}{DP}$	OUTSIDE DIAMETER	$OD = \frac{N - 2}{DP}$	CIRCULAR THICKNESS	$CT = \frac{CP}{2}$



CHORDAL THICKNESS AND CHORDAL ADDENDUM OF A GEAR TOOTH

MEASURING CHORDAL THICKNESS OF GEAR TOOTH



SPUR GEAR TERMS

ADDENDUM

WORKING DEPTH

WHOLE DEPTH

OUTSIDE DIA.

PITCH DIA.

PITCH DIA.

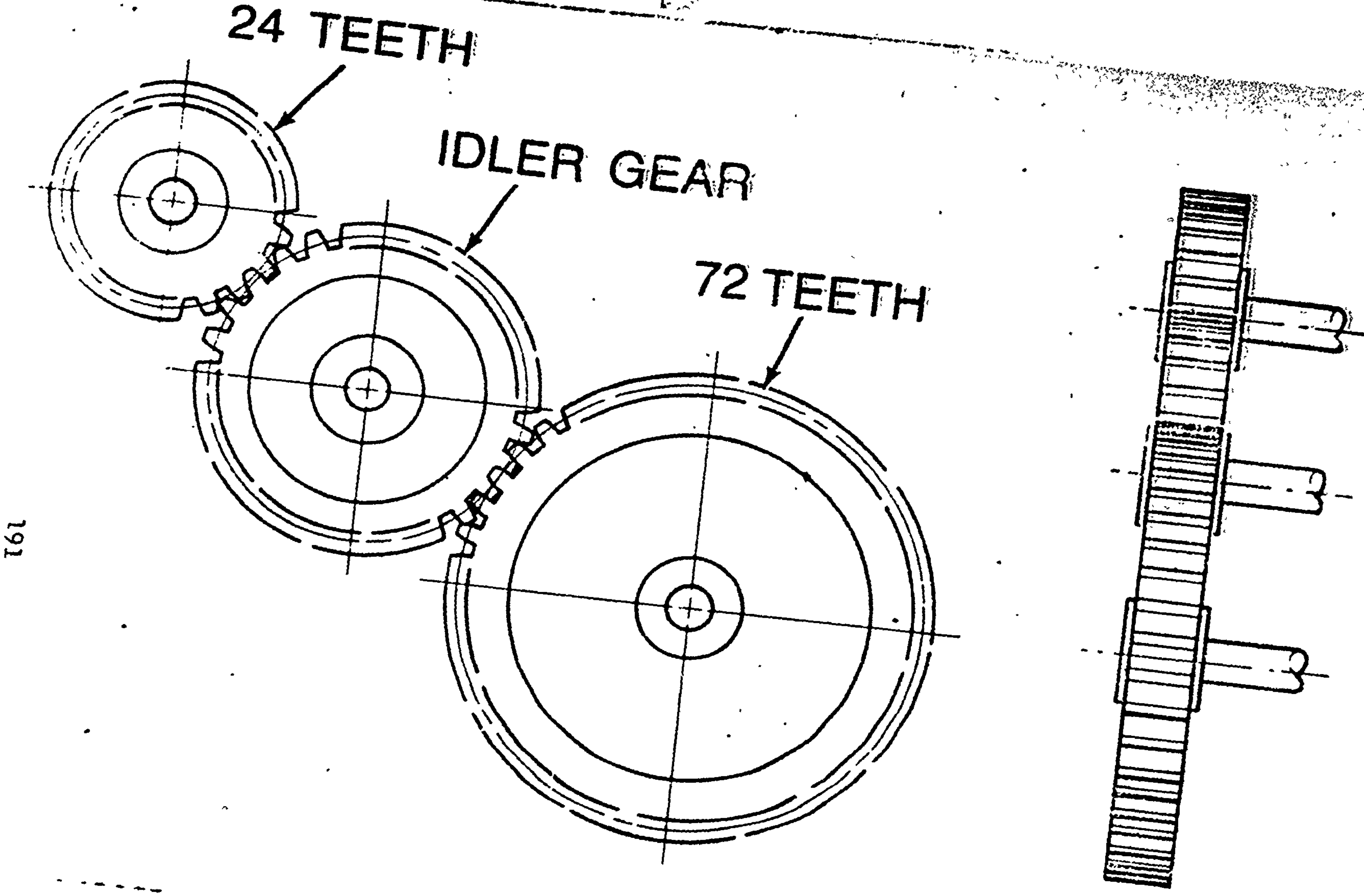
GEAR

PINION

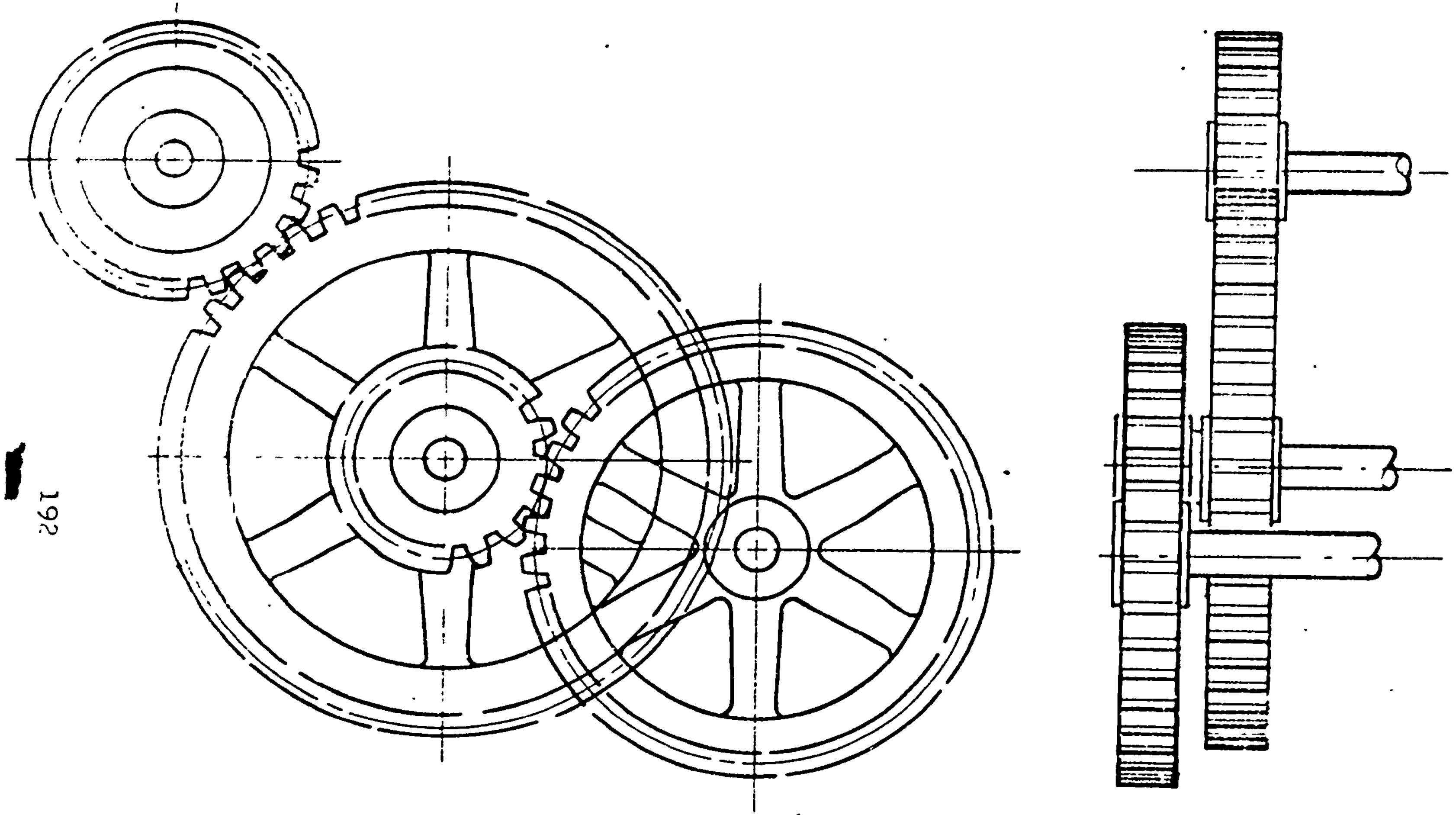
CLEARANCE

DEDENDUM

CENTRE DISTANCE



SIMPLE GEARING



COMPOUND GEARING

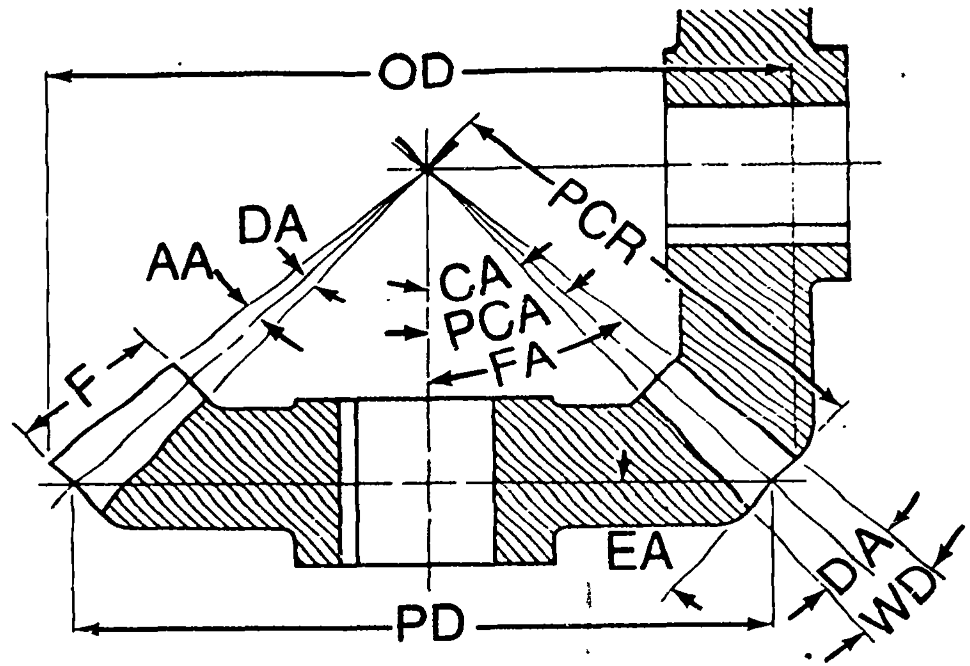
$$\frac{32 \times 24 \text{ (DRIVING GEARS)}}{64 \times 56 \text{ (DRIVEN GEARS)}}$$

32 TEETH

64 TEETH

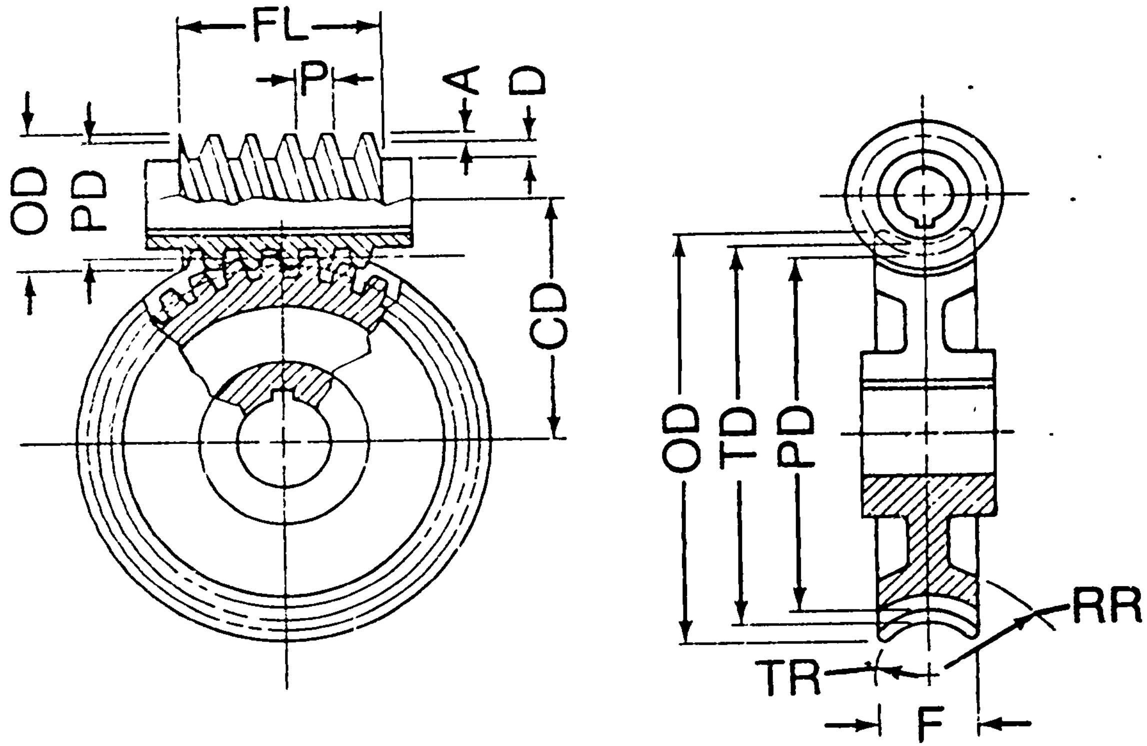
56 TEETH

24 TEETH



BEVEL GEAR NOMENCLATURE AND FORMULAE

PITCH CONE ANGLE	$TAN\ PCA_G = \frac{N_G}{N_P}$	EDGE ANGLE	EA = PCA
PITCH DIAMETER	$PD = \frac{N}{DP}$	FACE ANGLE	FA = PCA + AA
PITCH CONE RADIUS	$PCR = \frac{PD}{2\ SIN\ PCA}$	CUTTING ANGLE	CA = PCA − DA
ADDENDUM ANGLE	$TAN\ AA = \frac{A}{PCR}$	OUTSIDE DIAMETER	OD = PD − (2A COS PCA)
DEDENDUM ANGLE	$TAN\ DA = \frac{D}{PCR}$	FACE	F NOT TO EXCEED $\frac{1}{3}$ PCR



WORM GEAR NOMENCLATURE AND FORMULAE

FOR THE WORM		FOR THE WORM GEAR			
PITCH DIAMETER	$PD = OD - 2A$	PITCH DIAMETER	$PD = \frac{N}{DP}$	THROAT RADIUS	$TR = \frac{1}{2}PD_w - A$
OUTSIDE DIAMETER	$OD = PD + 2A$	THROAT DIAMETER	$TD = PD + 2A$	RIM RADIUS	$RR = \frac{1}{2}\,'D_w + CP$
CENTRE DISTANCE	$CD = \frac{PD_w + PD_G}{2}$	OUTSIDE DIAMETER	$OD = TD + .4775$	FACE	$F = 2.38CP + .25$

ASSIGNMENT SHEET

TITLE: SPUR GEAR PROBLEMS

UNIT: GEARS

OCCUPATION: MACHINIST

OBJECTIVE: To give the student practice in solving spur gear problems.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 13, pages 313-328.

QUESTIONS:

1. What is the pitch diameter of a 6 pitch gear with 25 teeth?

2. A gear has 36 teeth and a pitch of 6. What is the pitch diameter.

3. How many teeth are there on a 20 pitch gear 5 inches in diameter?

4. The pitch of a gear is 24 and the diameter of the gear blank is 6.750 inches. How many teeth will it have when cut?

5. A 10 pitch gear has a pitch diameter of 5.750 inches. What is the O. D.?

6. What is the clearance of a 12 pitch gear?

7. What is the addendum of a 6 pitch gear?

8. What is the dedendum of a 10 pitch gear?

9. What is the working depth of a 6 pitch gear?

10. What is the whole depth of a 6 pitch gear?

11. Find the center to center distance of a 10 pitch 18 tooth gear and a 10 pitch 24 tooth gear.

12. What is the circular pitch of a 8 pitch 24 tooth gear?

13. What is the linear pitch of a 8 pitch 32 tooth rack?

14. What is the pitch of a 32 tooth gear 3 inches in diameter?

15. A stripped gear had 38 teeth. As close as could be measured, the O. D. was 3.992 inches. Find the correct O. D. and pitch for a new gear.

BEST COPY AVAILABLE

ASSIGNMENT SHEET

TLE: ABRASIVES AND ABRASIVE PRODUCTS

IT: ABRASIVES AND GRINDING

CUPATION: MACHINIST

JECTIVE: To acquaint the student with the common types of abrasives and abrasive products.

FERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 14.

RECTIONS: Read the above reference and answer the following questions.

ESTIONS:

1. What are the most commonly used natural abrasives?

2. What are the two most used manufactured abrasives?

3. What abrasive is used for most grinding wheels used for grinding steel?

4. How is the grain size of abrasives classified?

5. What are the five types of bonds used in making grinding wheels?

6. Which bond is used on most griding wheels?

7. What is the structure of a grinding wheel?

8. What is the grade of bond?

9. On what types of material should silicon carbide abrasives be used?

10. On what types of material should aluminum oxide abrasives be used?

11. What factors govern the grinding wheel selection for a given job?

12. Explain the marking system used on grinding wheels.

13. What is the difference between dressing and trueing a grinding wheel?

14. List the abrasive products other than grinding wheels.

15. List the rules that apply to the care and handling of grinding wheels and other abrasive products.

Sequence Prefix

Symbol	Abrasive Type	Grain Size	Grade	Structure	Bond Type	Manufactor's Record

Typical Marking: 51 - A - 36 - L - 5 - V - 23

…ufactor's symbol indicating …t kind of …rasive (use is …ional)

Aluminum Oxide

- Silicon carbide

Coarse	Medium	Fine	Very Fine
10	30	70	220
12	36	80	240
14	46	90	280
16	54	100	320
20	60	120	400
24		150	500
		180	600

Dense to Open

1-Dense	9
2	10
3	11
4	12
5	13
6	14
7	15-Open
8	etc.

(use is Opitional)

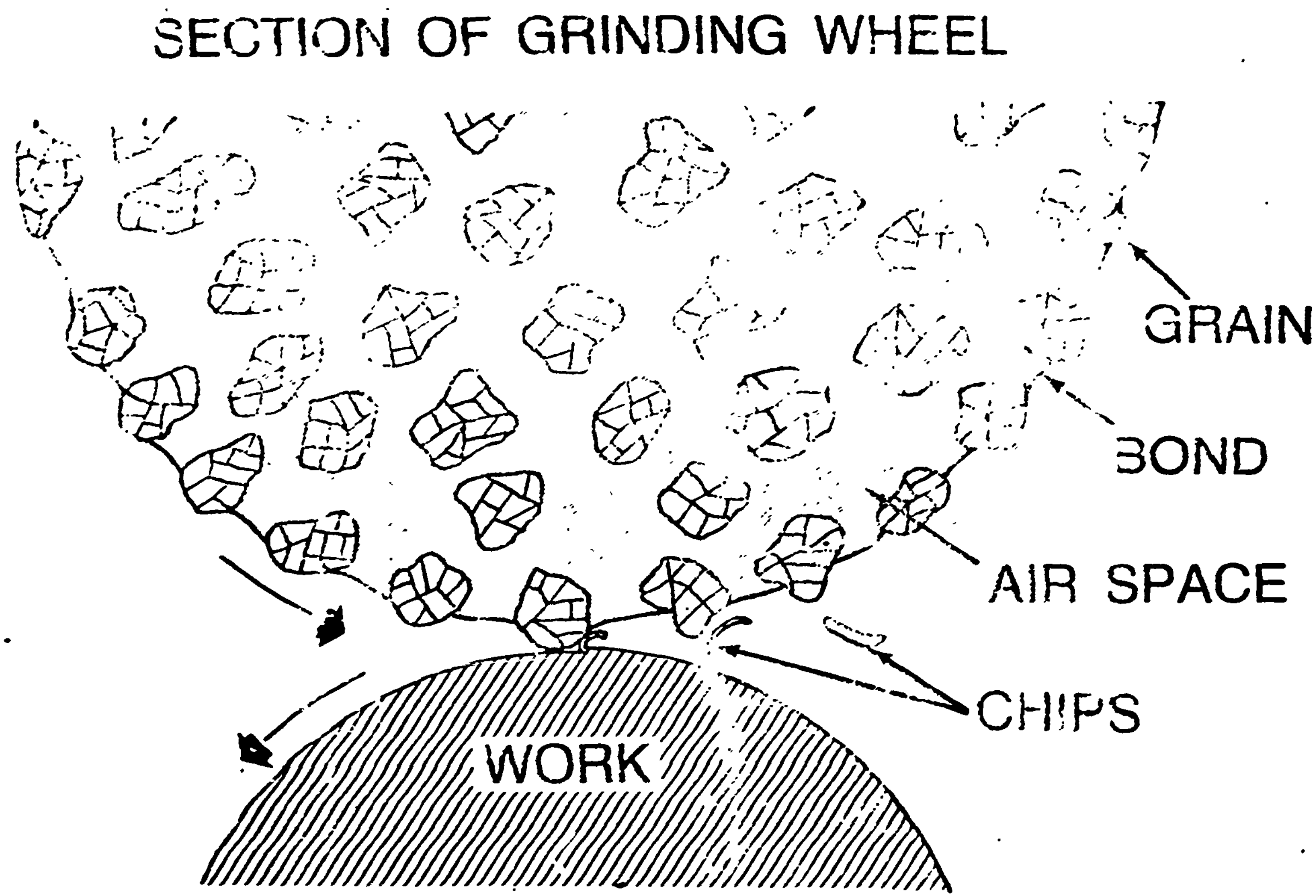
Manufactor's private marking to identify wheel (use is optional)

Vitrified - V

Silicate - S
Rubber - R
Resinoid - B
Shellac - E
Oxychloride - O

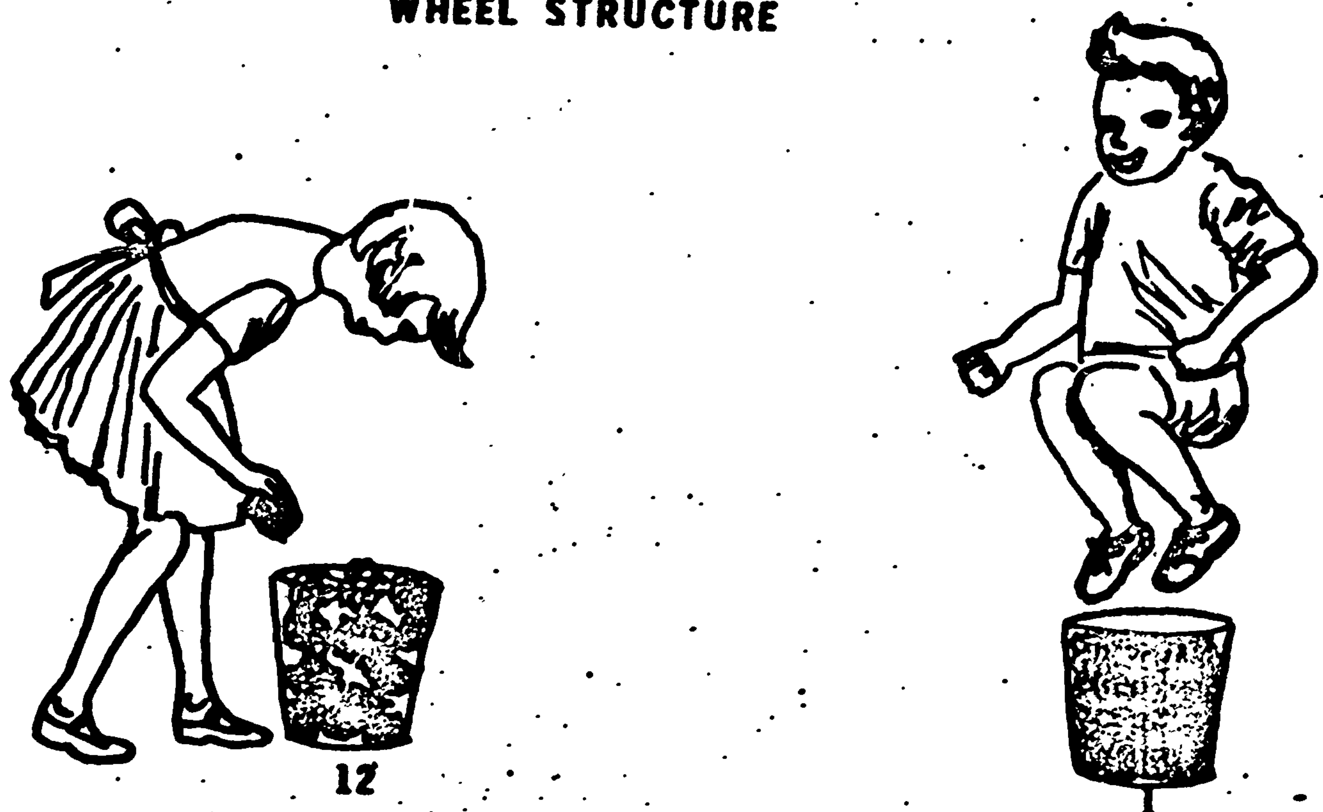
…t Medium Hard

…DEFGHIJK LMNOPQRSTUVWXYZ



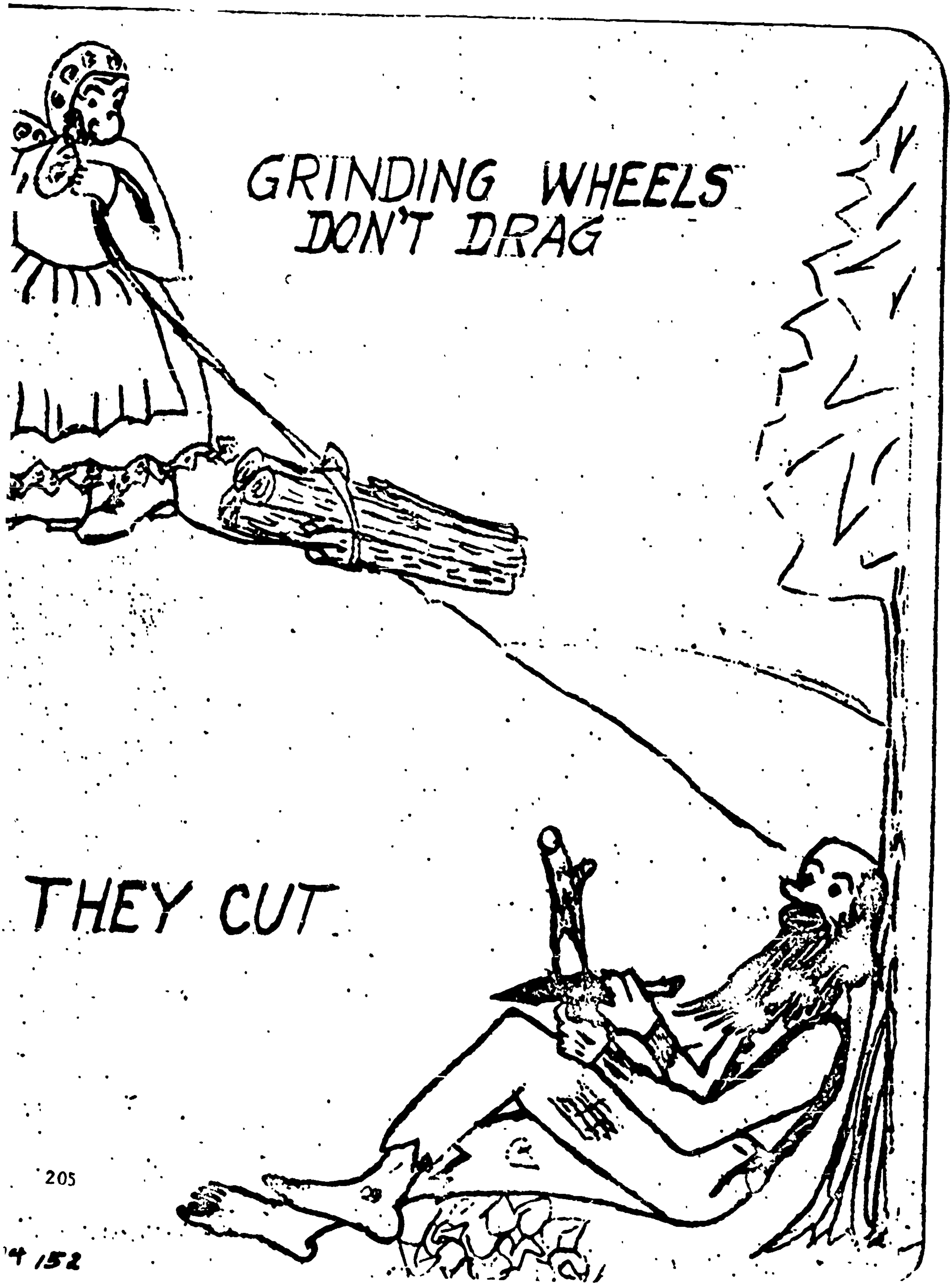
CUTTING ACTION OF A GRINDING WHEEL

WHEEL STRUCTURE



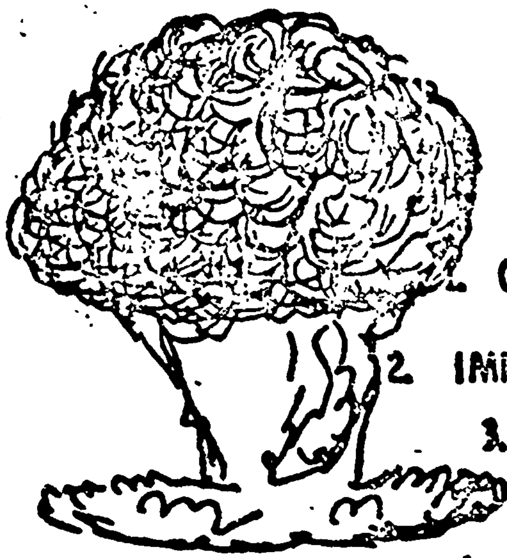
12

SAME SIZE GRAIN - JUST SPACED DIFFERENTLY

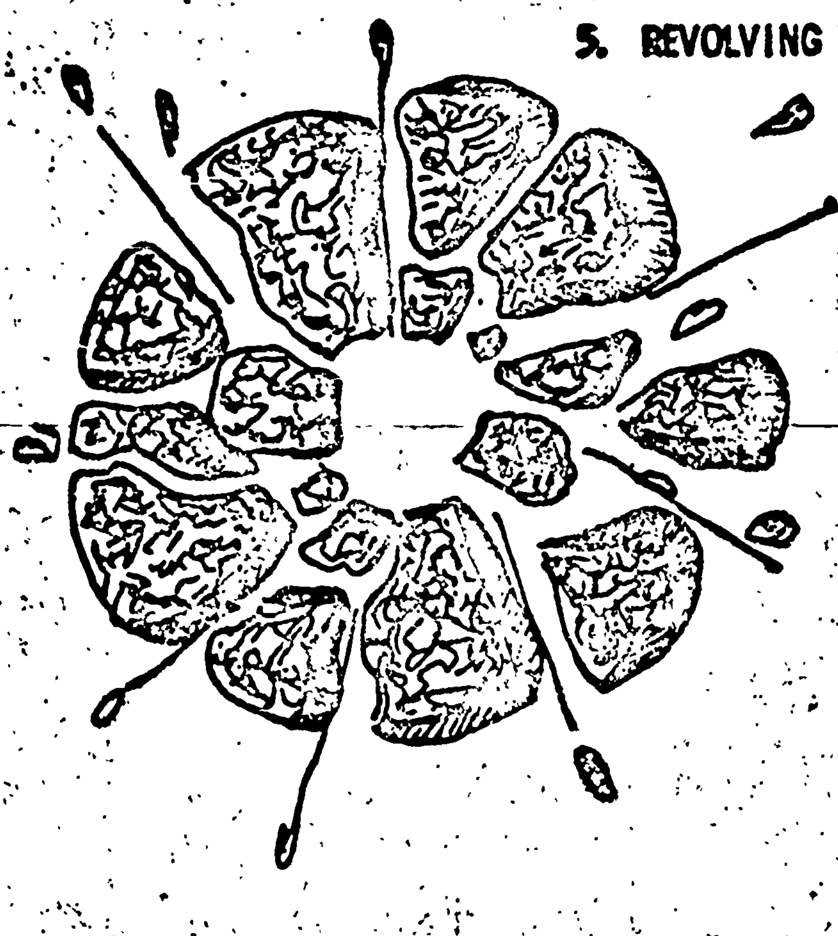


4 152

GRINDING WHEELS EXPLODE



1. CRACKED WHEELS
2. IMPROPER WHEEL MOUNTING
3. UNBALANCED GRINDING WHEELS
4. IMPROPER WHEEL USE
5. REVOLVING TOO FAST



ASSIGNMENT SHEET

TITLE: SURFACE GRINDERS AND GRINDING OPERATIONS

UNIT: ABRASIVES AND GRINDING

OCCUPATION: MACHINIST

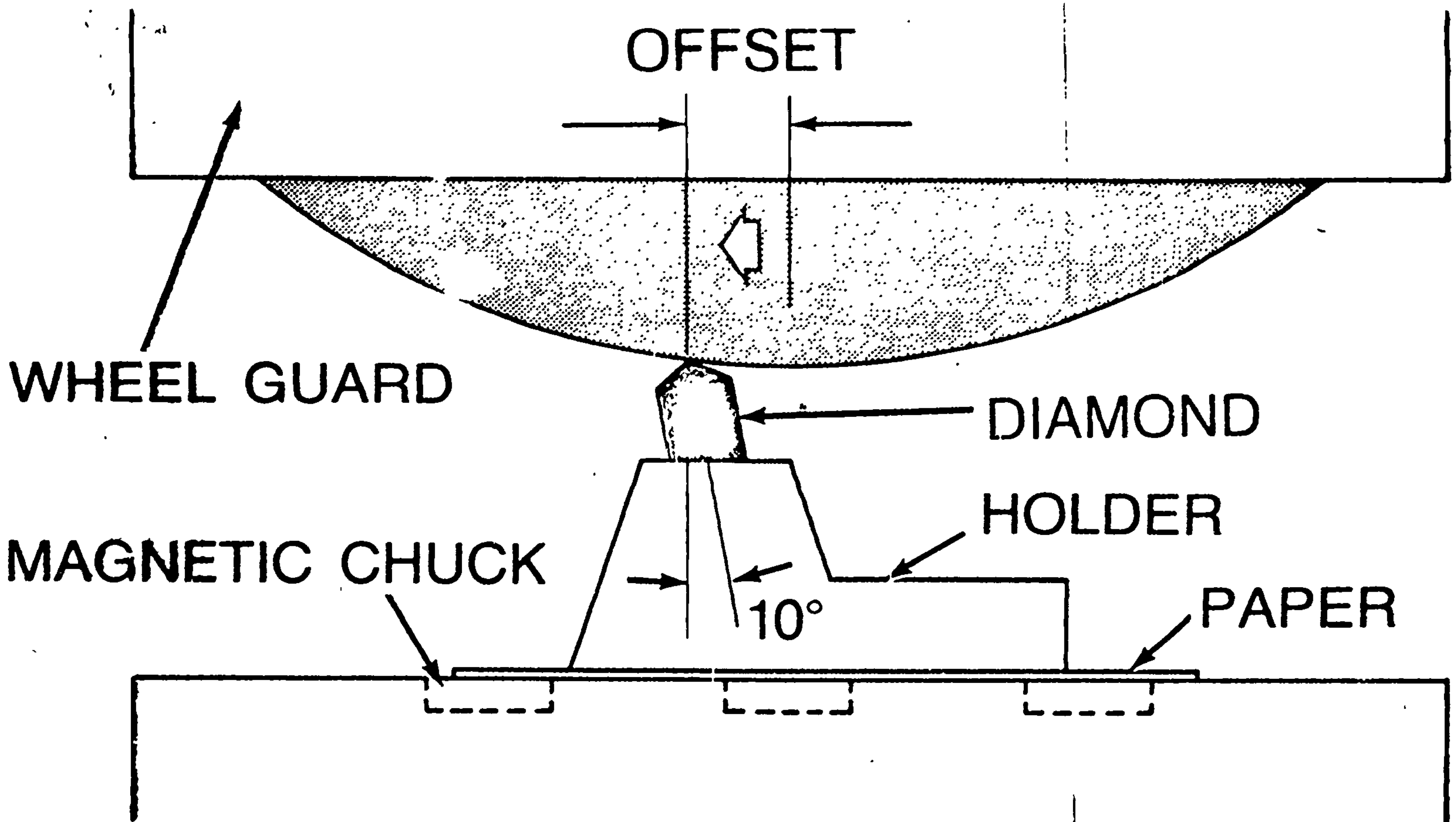
OBJECTIVE: To acquaint the student with the types and use of surface grinders.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 15, pages 347-364.

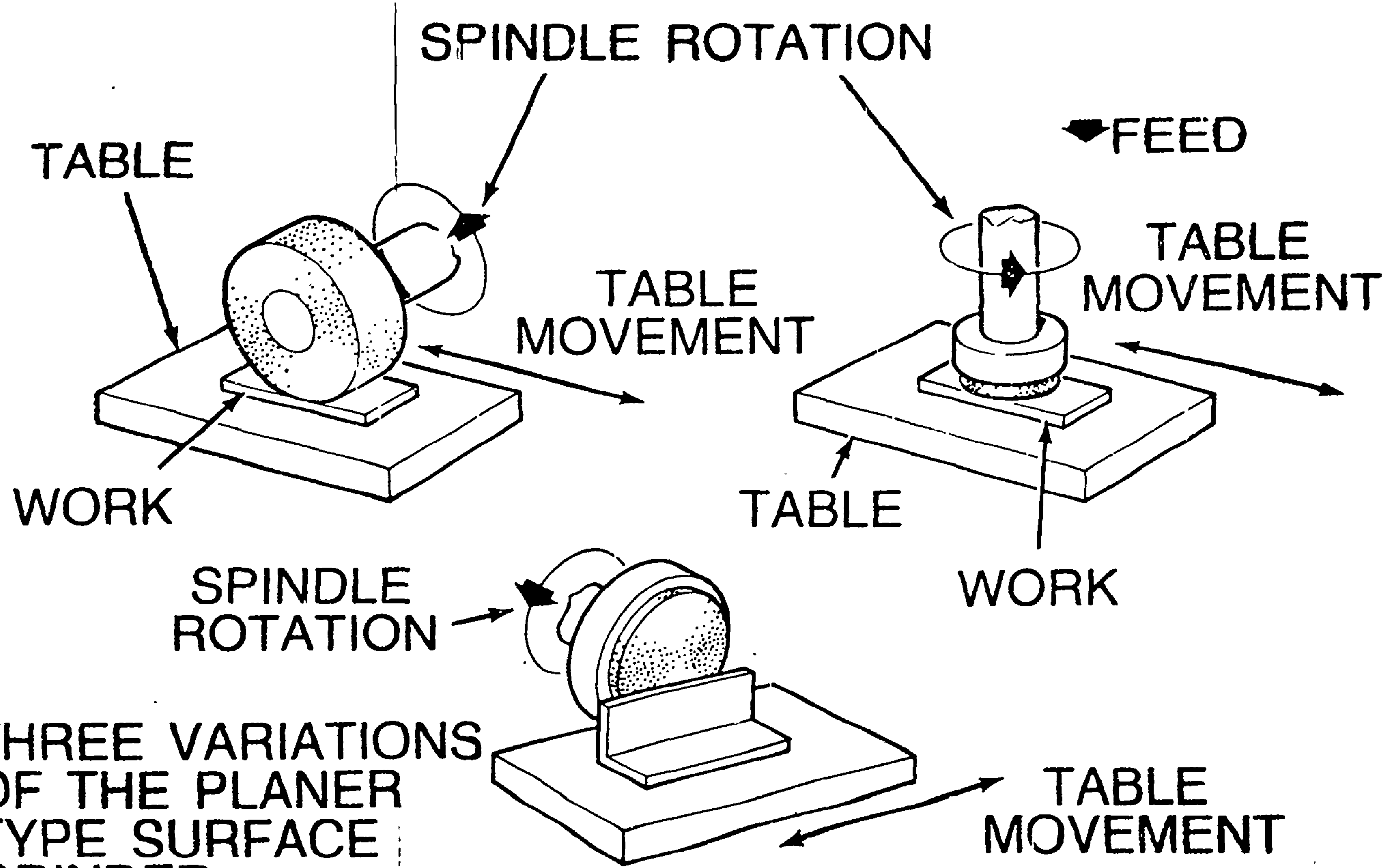
DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. What is the difference between the planer and rotary type surface grinders?
2. What wheel shapes are commonly used on surface grinders?
3. How may wheels be checked for cracks?
4. What is the purpose of blotting - paper washers on the sides of the wheel?
5. Why is it necessary to keep surface grinder wheels sharp and clean?
6. What is used to dress and true surface grinding wheels?
7. How is work usually held on a surface grinder?
8. Explain the process for mounting a wheel on a surface grinder.
9. Why should one stand to one side when a grinder is turned on after a new wheel has been mounted?
10. What safety rules apply to the operation of a surface grinder?

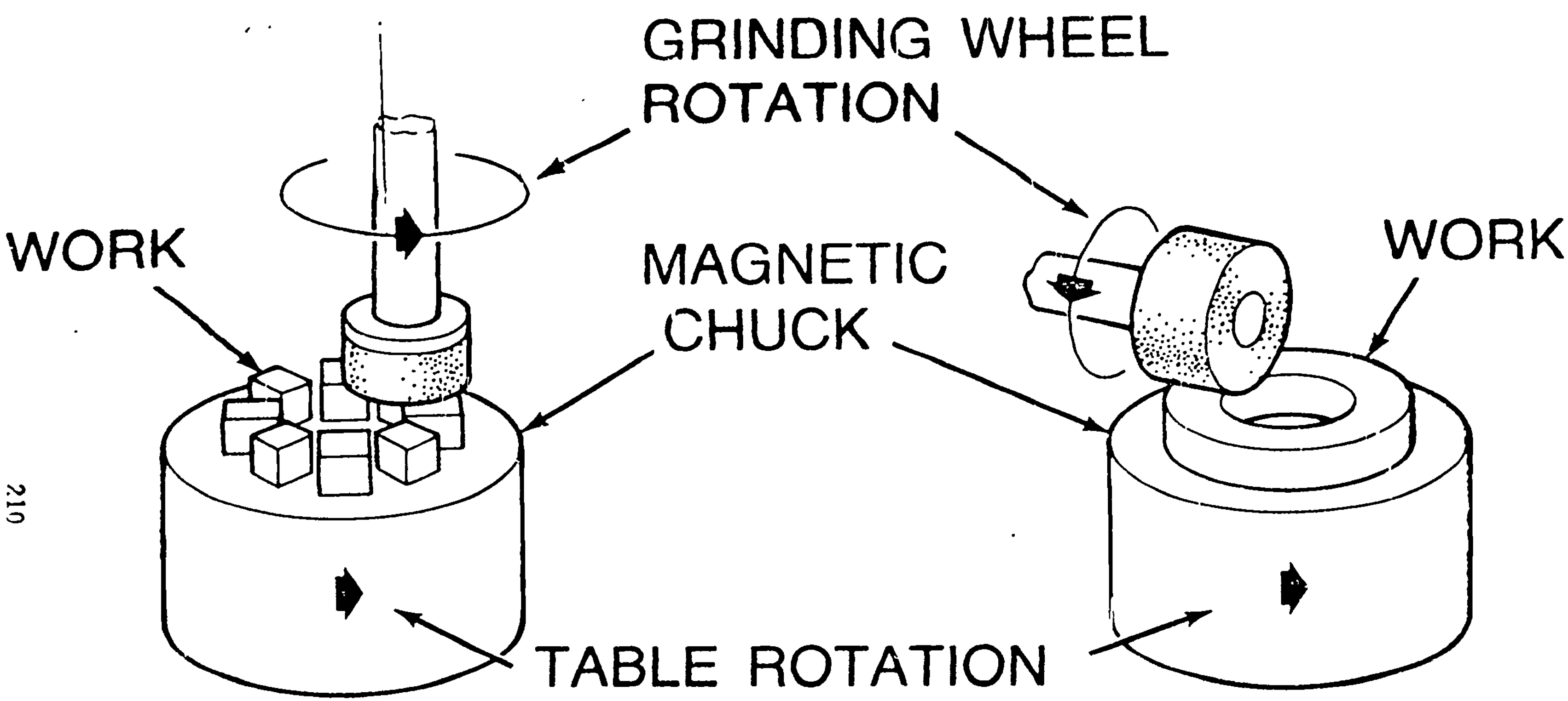


DRESSING A SURFACE GRINDING WHEEL



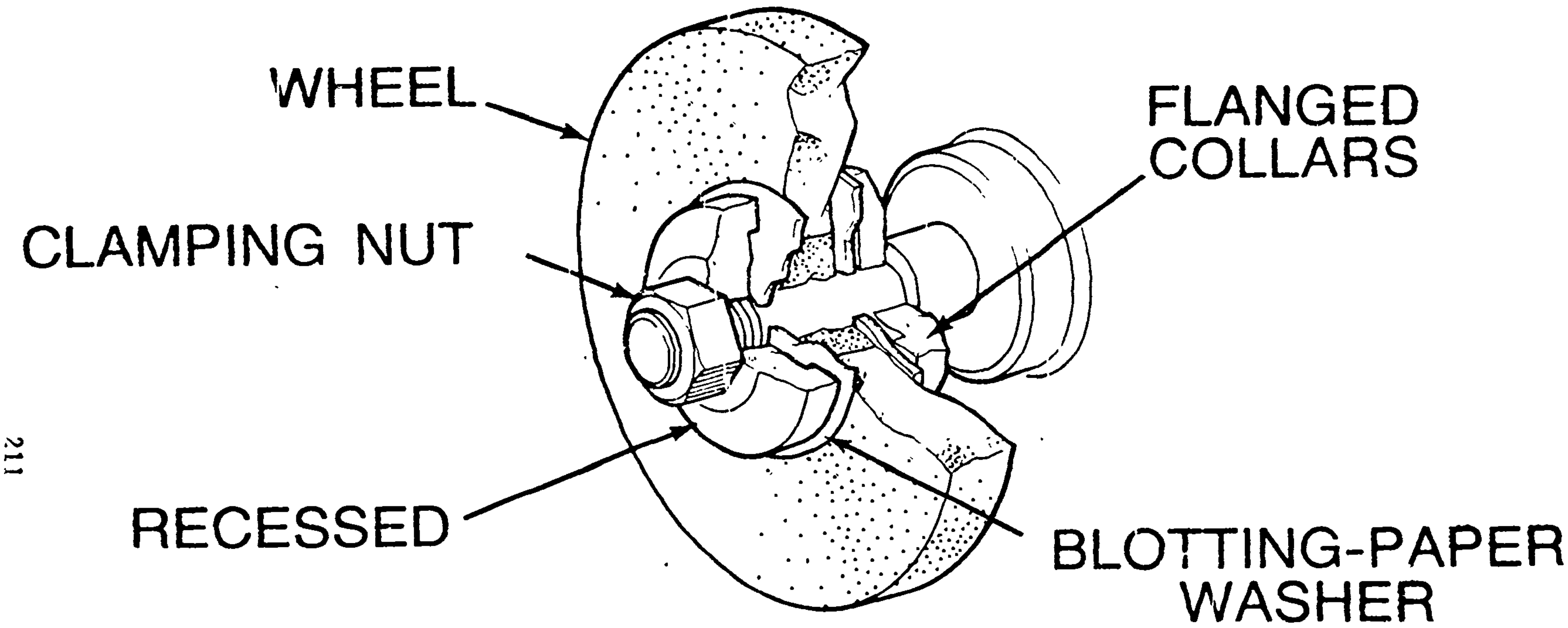
THREE VARIATIONS OF THE PLANER TYPE SURFACE GRINDER

TYPES OF SURFACE GRINDING. NO. 1



TWO VARIATIONS OF THE ROTARY
TYPE SURFACE GRINDER

TYPES OF SURFACE GRINDING. NO. 2



MOUNTING A GRINDING WHEEL ON A STRAIGHT SPINDLE

ASSIGNMENT SHEET

TITLE: CYLINDRICAL GRINDING AND GRINDING OPERATIONS

UNIT: ABRASIVES AND GRINDING

OCCUPATION: MACHINIST

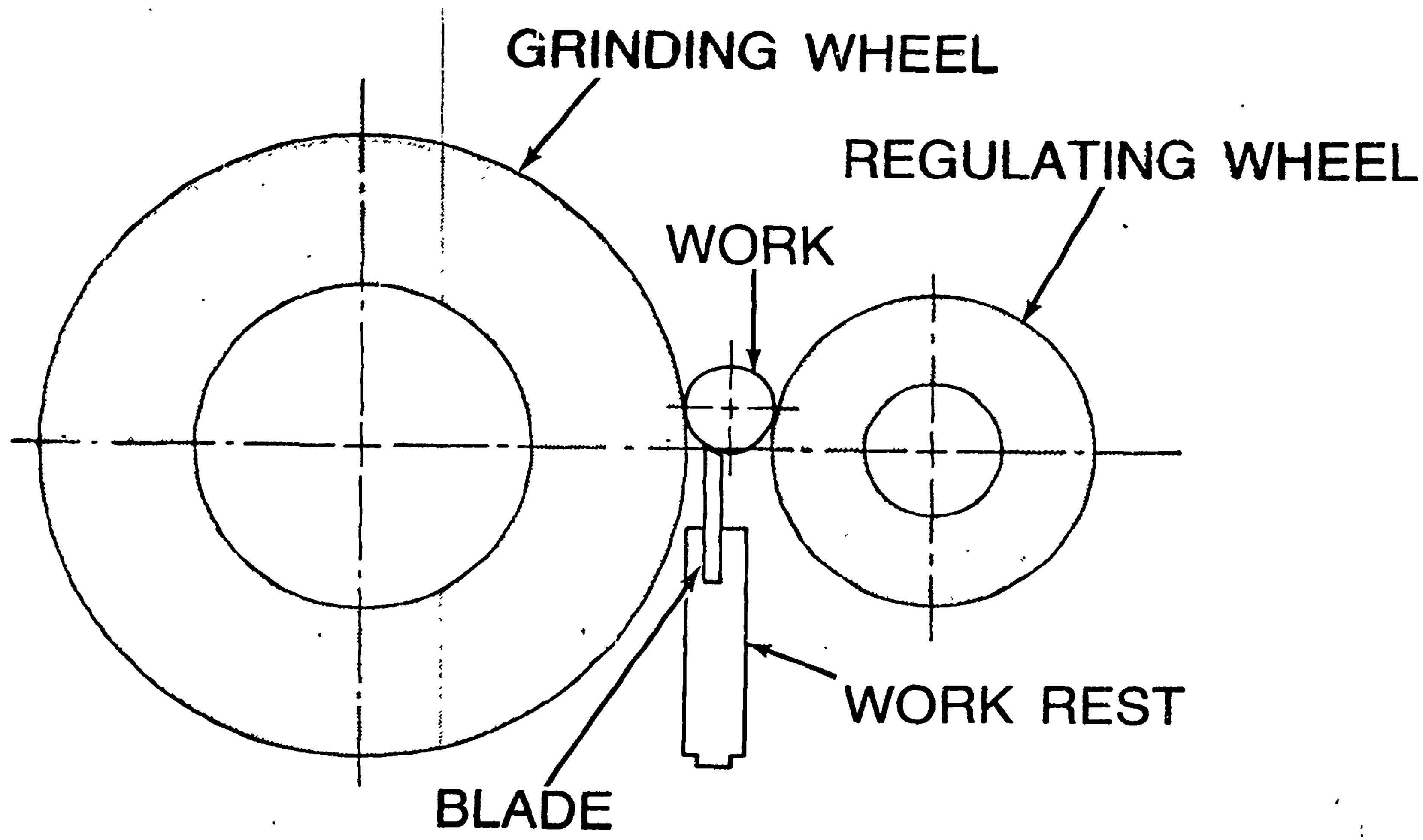
OBJECTIVE: To acquaint the student with the types and use of cylindrical grinders.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 15, 364-386.

DIRECTION: Read the above reference and answer the following questions.

QUESTIONS:

1. What is the purpose of external grinding?
2. Into what three major groups are external grinders divided?
3. What movements are important in any cylindrical grinder?
4. What happens to the job if the grinding wheel completely over-runs the end of the work?
5. Why is it not possible to have a set rule for the rate of work speed?
6. How can tapering of the work be overcome?
7. How should the diamond be used to make the wheel fast cutting?
8. How is it possible to produce taper work on the cylindrical grinder?
9. What is meant by a universal tool grinder?
10. What kind of work can be done on the centerless grinder?



CENTRELESS GRINDING—PRIMARY ELEMENTS

ASSIGNMENT SHEET

TITLE: GRINDERS FOR CUTTING TOOLS AND THREAD GRINDERS

UNIT: ABRASIVES AND GRINDING

OCCUPATION: MACHINIST

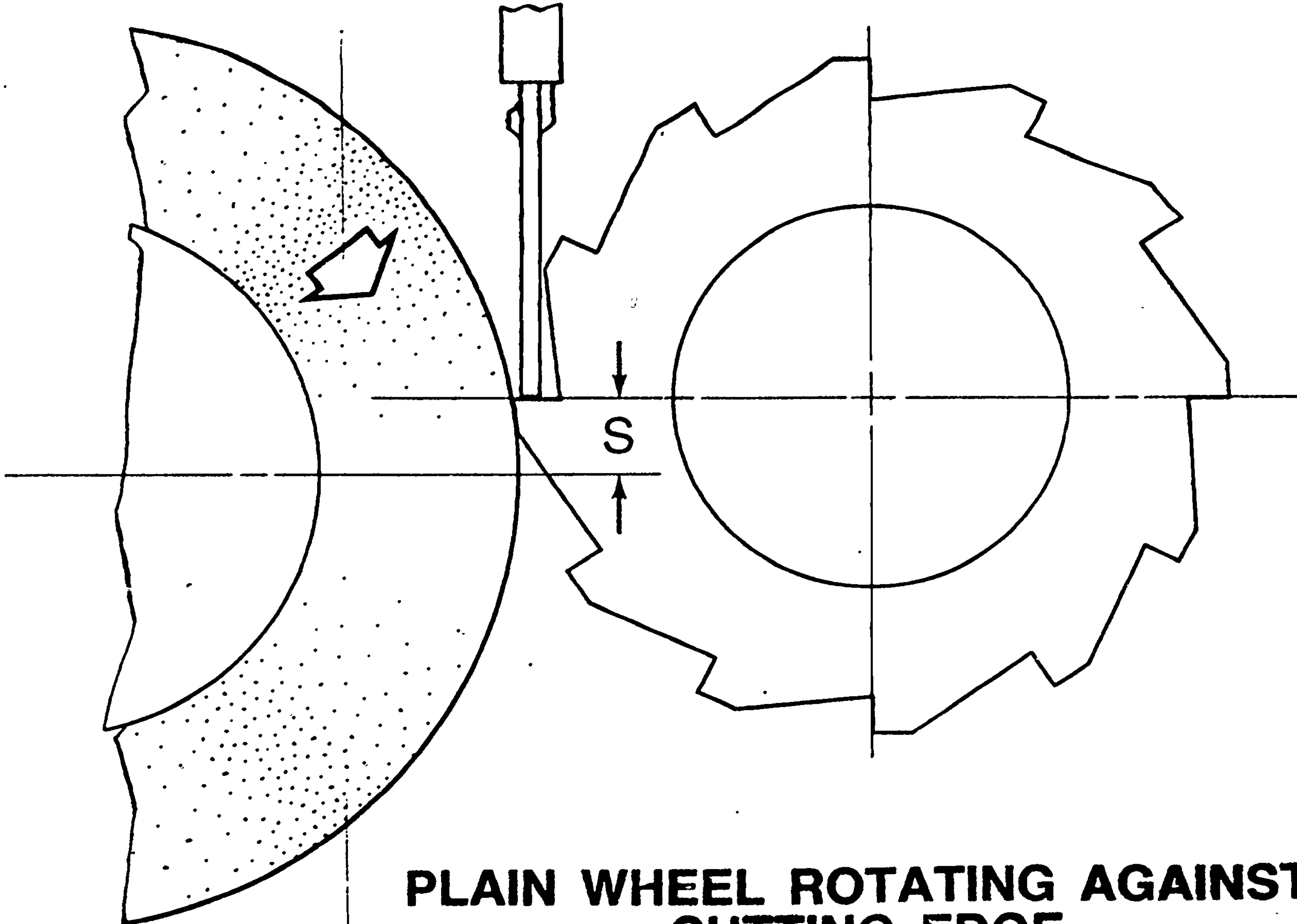
OBJECTIVE: To acquaint the student with the types of grinders for sharpening cutters and grinding threads.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 15, pages 387-399.

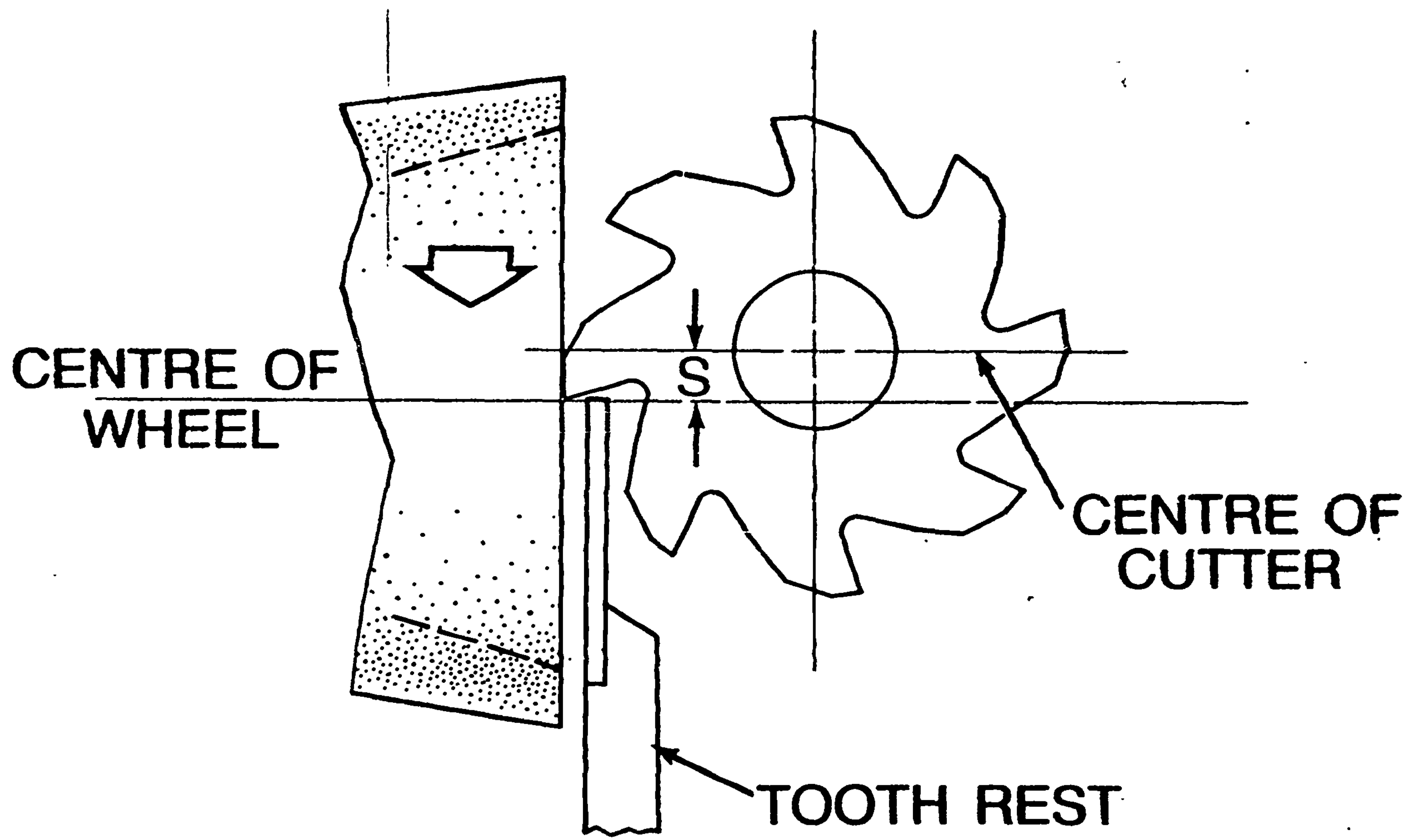
DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

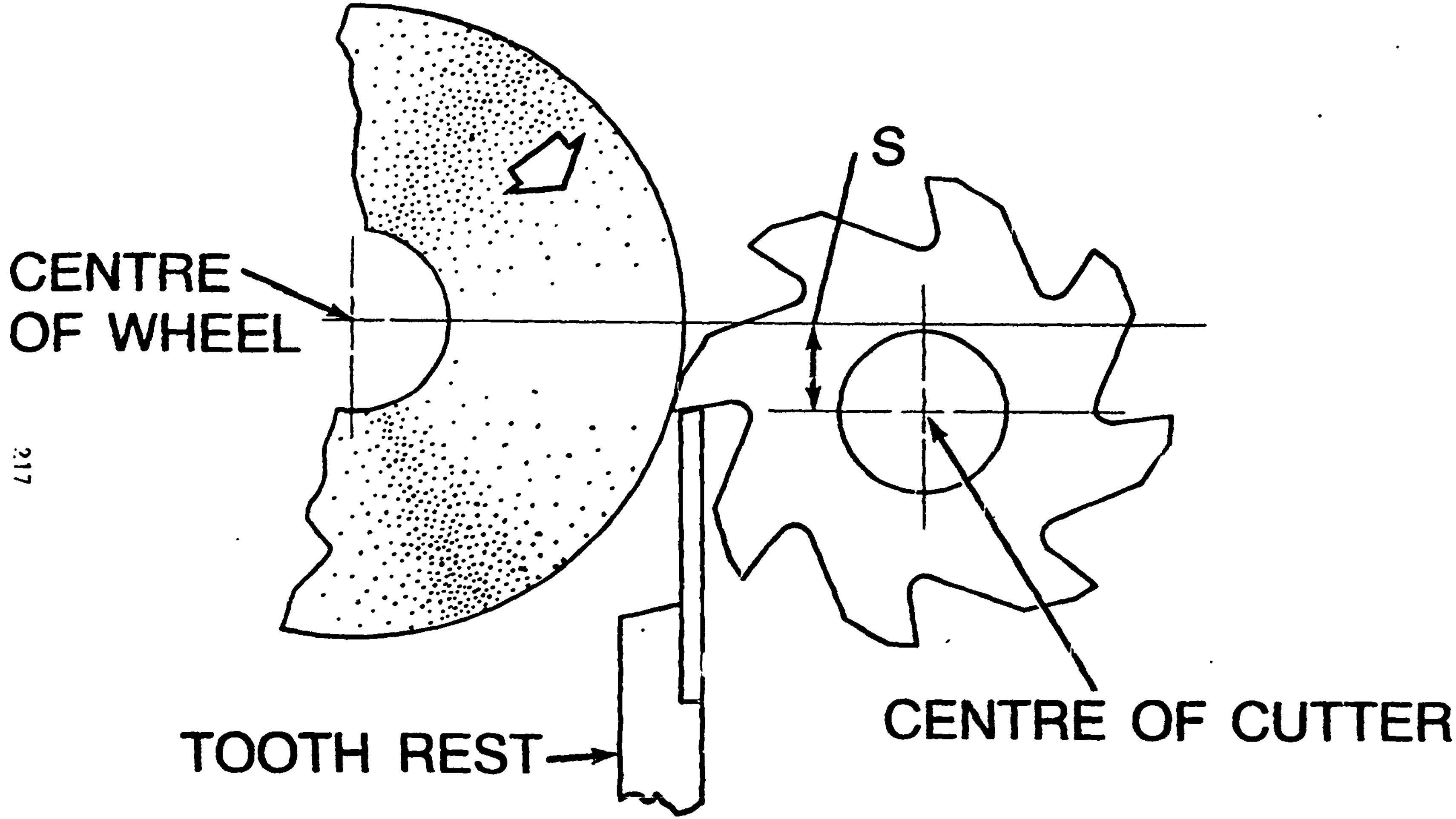
1. Explain the difference between a floor and bench grinder.
2. The bench grinder is used for what general type of work?
3. What type of grinding wheels are suitable for general-purpose cutter grinding?
4. What shapes of wheels are commonly used for cutter grinding?
5. What causes burning or cutting edges in cutter grinding?
6. Name two types of tooth rest most generally used.
7. How are end mills generally ground?
8. List six factors to remember when sharpening cutters.
9. Explain the difference between and the capacity of the internal and external thread grinder.
10. How are thread grinding wheels dressed?



PLAIN WHEEL ROTATING AGAINST CUTTING EDGE

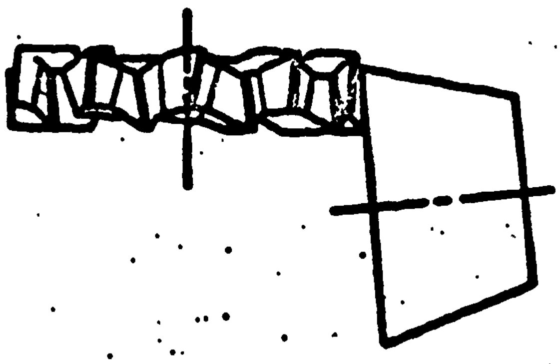


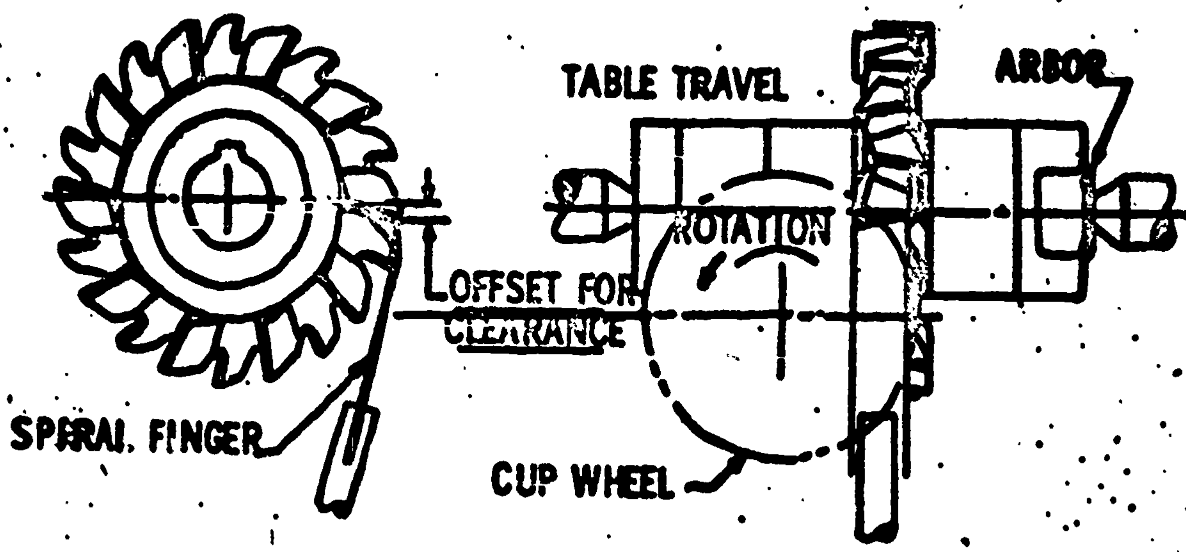
CUP WHEEL AWAY FROM CUTTING EDGE



PLAIN WHEEL AWAY FROM CUTTING EDGE

SHARPENING OF ALTERNATE TOOTH MILLING CUTTERS





ASSIGNMENT SHEET

TITLE: HEAT-TREATMENT OF STEEL

UNIT: HEAT-TREATING

OCCUPATION: MACHINIST

OBJECTIVE: To acquaint the student with the heat-treating processes used on various types of steel.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 16, pages 400-420.

QUESTIONS:

1. What is steel?
2. What are three grades of carbon steel?
3. How much carbon must be present in steel before it can be hardened noticeably?
4. What are alloy steels?
5. What are the most common methods used to determine the hardness of steel?
6. How is the temperature of a head treat furnace controlled?
7. What is the SAE system of classifying steeel?
8. Explain how a piece of high carbon steel is hardened?
9. Why is tool steel drawn or tempered after it is hardened?
10. How long should a piece of steel be left in a heat treat furnace?

TERMS DEFINE:

1. Alloy
2. Straight carbon steel
3. Hardened
4. Critical Temperature
5. Draw
6. Temper
7. Anneal

ASSIGNMENT SHEET

8. Quened

9. Case harden

10. Carburizing

11. Normalize

12. Stress relieve

13. Cyaniding

14. Flame hardening

15. High speed steel

ASSIGNMENT SHEET

TITLE: SURFACE FINISHES AND MEASUREMENT

UNIT: SURFACE FINISH

OCCUPATION: MACHINIST

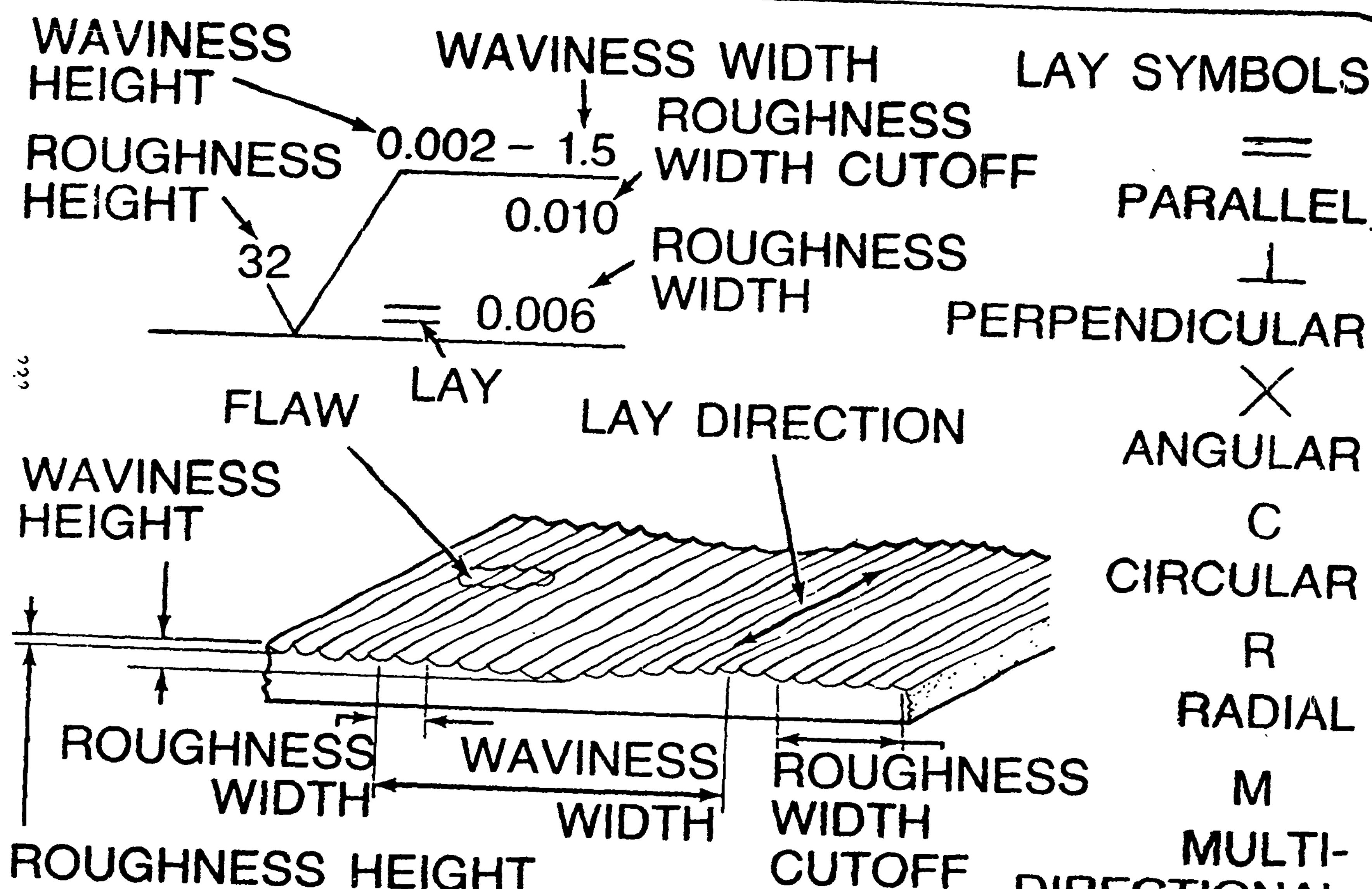
OBJECTIVE: To acquaint the student with surface finishes and their concern to the machinist.

REFERENCE: Anderson-Tatro. Shop Theory. New York McGraw Hill Book Co., Inc. Chapter 17 pages 421-432.

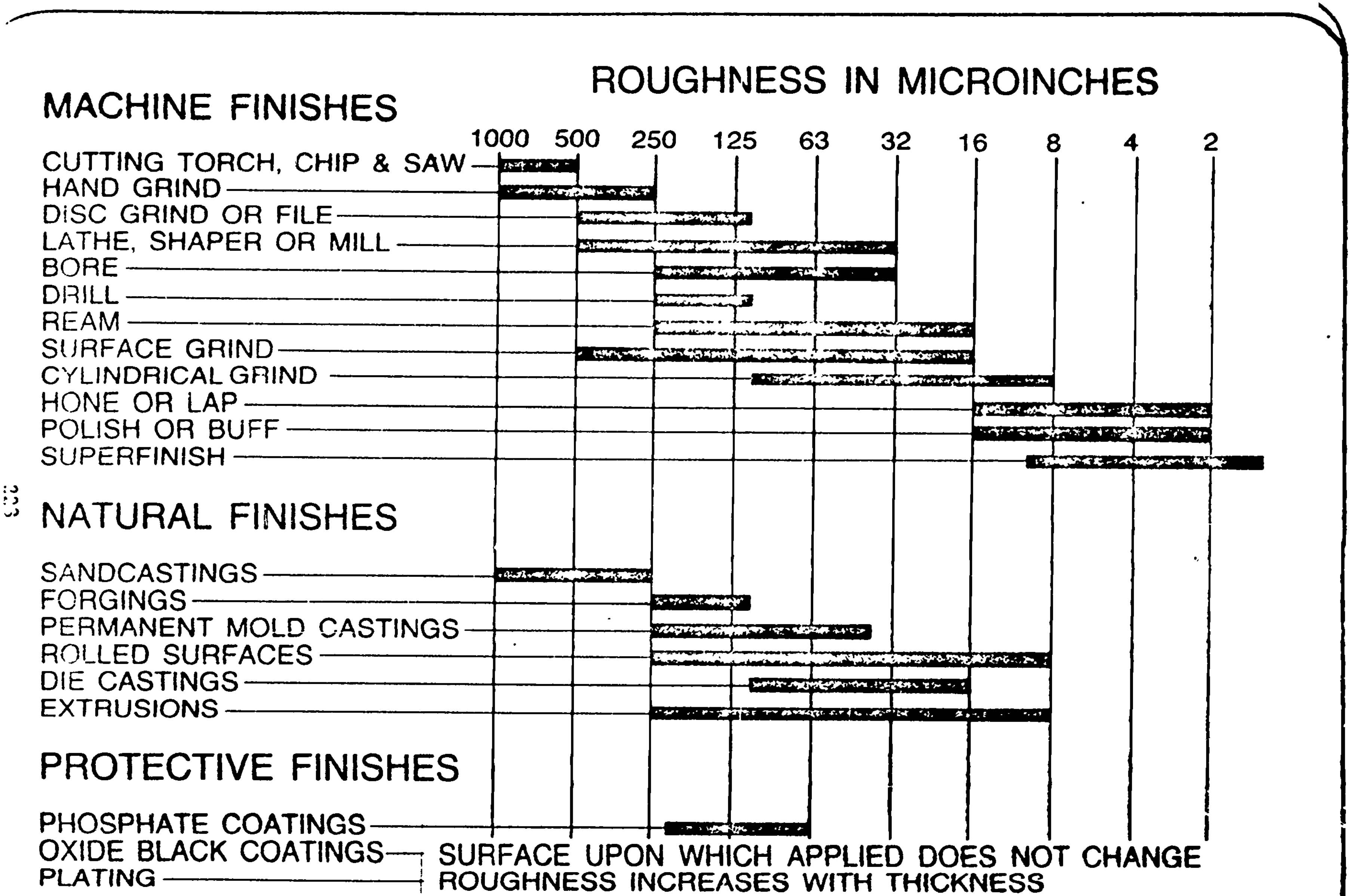
DIRECTIONS: Read the above reference and answer the following questions.

QUESTIONS:

1. What is meant by surface finish?
2. What factors contribute to the quality of a surface finish?
3. How is the degree of surface roughness measured?
4. Why are jobs checked for the quality of surface finish during production.
5. How are the characteristics of a surface specified on a machine drawing.
6. Name the simplest way of judging the roughness of a surface.
7. What instruments may be used to check the surface finish?
8. What is a microinch?
9. How are the characteristics of a surface specified on a machine drawing?
10. What causes the irregularities in the surface of a machined job?



SURFACE FINISH SYMBOLS



SURFACE FINISHES

ASSIGNMENT SHEET

TITLE: POWER SAWS AND SAWING

UNIT: BAND MACHINING

OCCUPATION: MACHINIST

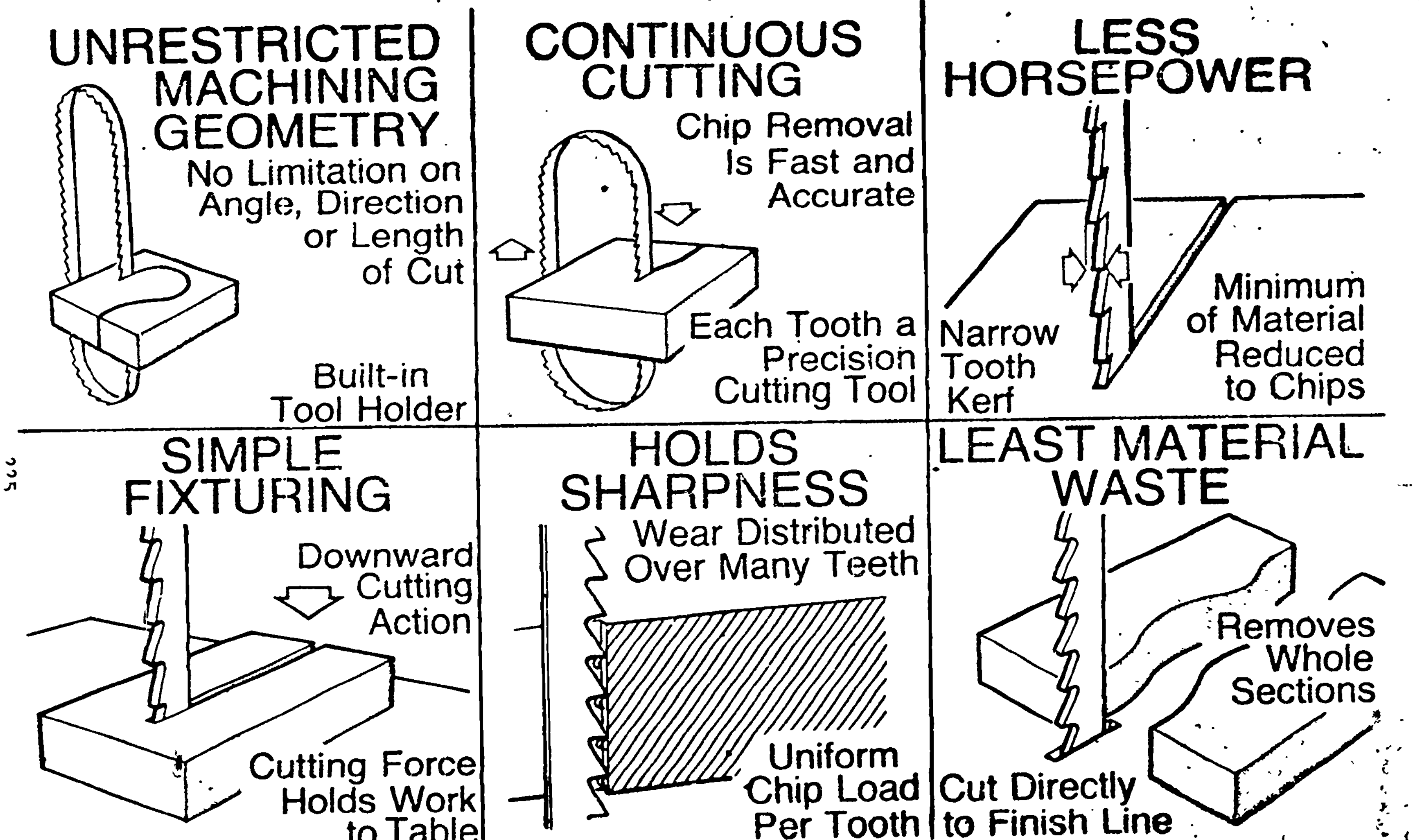
OBJECTIVE: To familarize the student with the types and use of power saws.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 18, pages 433-459.

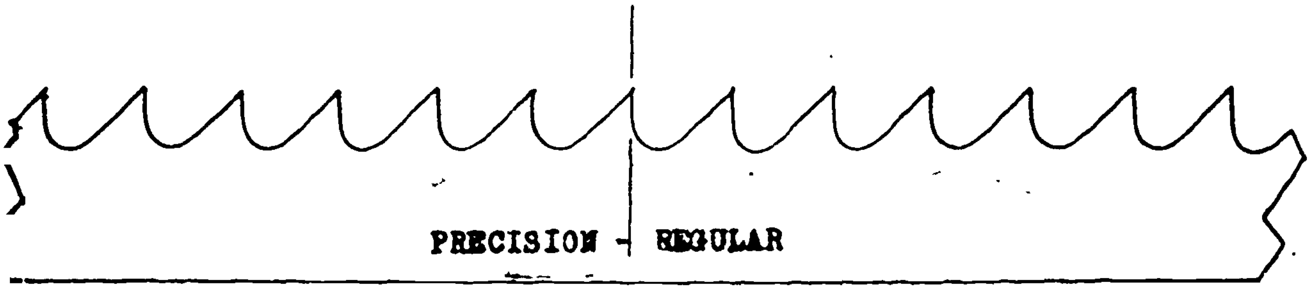
DIRECTIONS: Read the above reference and answer the following questions.

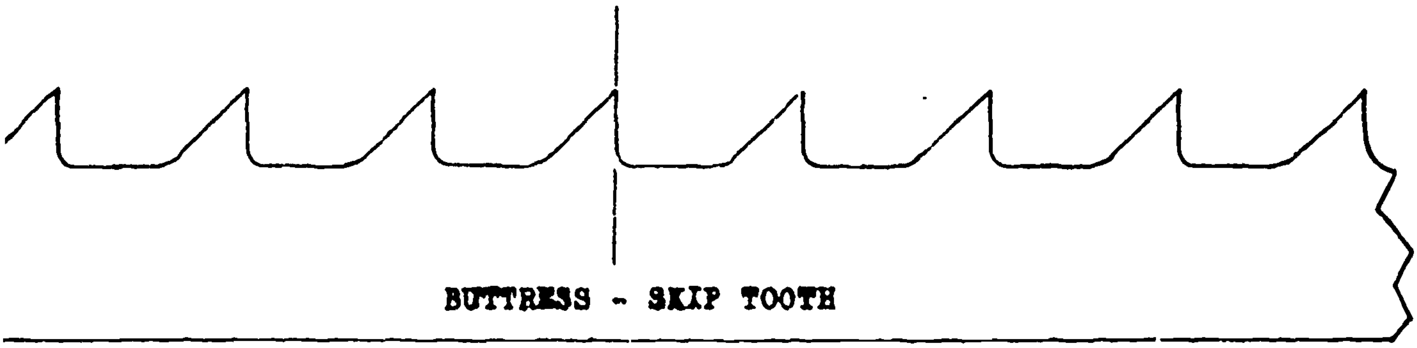
QUESTIONS:

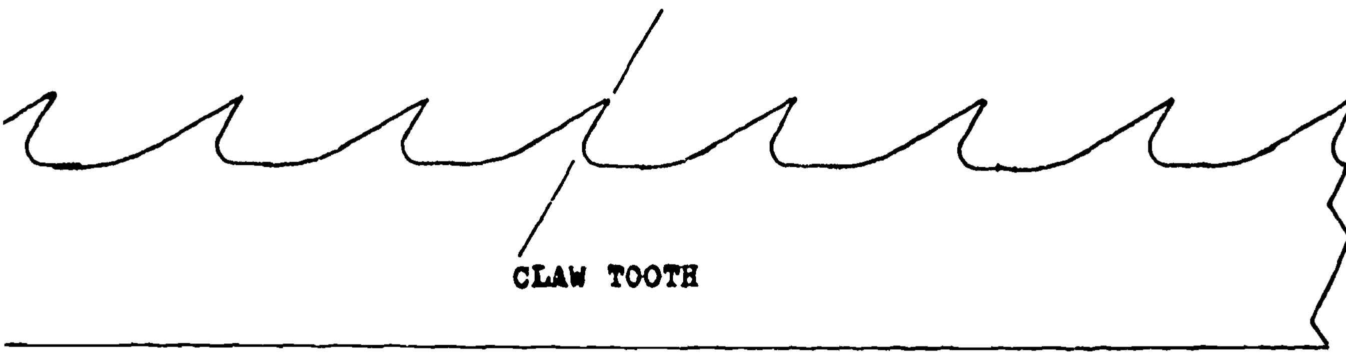
1. How many types of band saws are in use?
2. Why is the horizontal band saw more efficient than the reciprocating power saw?
3. How is work held in a power hacksaw?
4. What are the advantages of a band type power hacksaw?
5. How is the cutting speed of a band saw expressed?
6. Why are proper cuttin speeds important when using a metal-cutting band saw?
7. Why must a weld in a band saw blade be annealed?
8. What is friction sawing?
9. What determines the pitch blade to use?
10. What changes are necessary to convert the sawing machine into a filing machine?

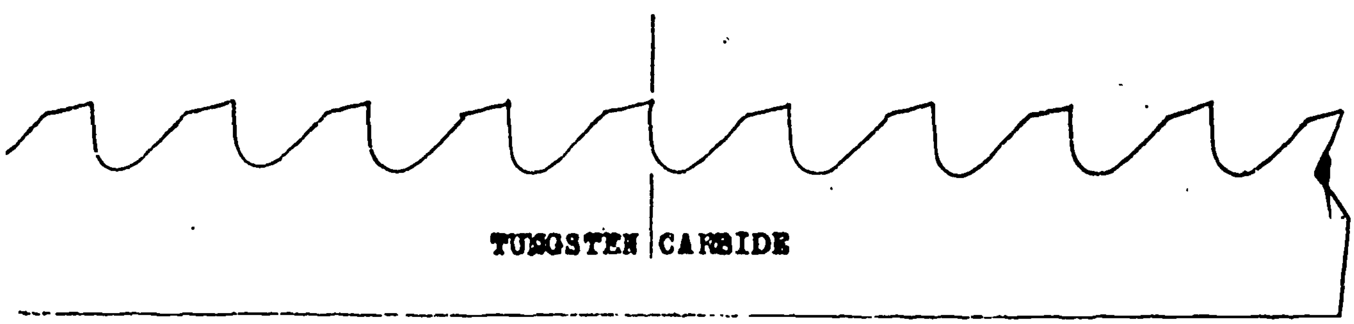


ADVANTAGES OF A CONTOUR CUTTING BAND SAW


PRECISION - REGULAR


BUTTRESS - SKIP TOOTH


CLAW TOOTH


TUNGSTEN CARBIDE

BEST COPY AVAILABLE

3. WIDTH

ALWAYS USE WIDEST BLADE:

(A) AVAILABLE IN DESIRED PITCH

(B) THAT WILL CUT THE SMALLEST RADIUS REQUIRED

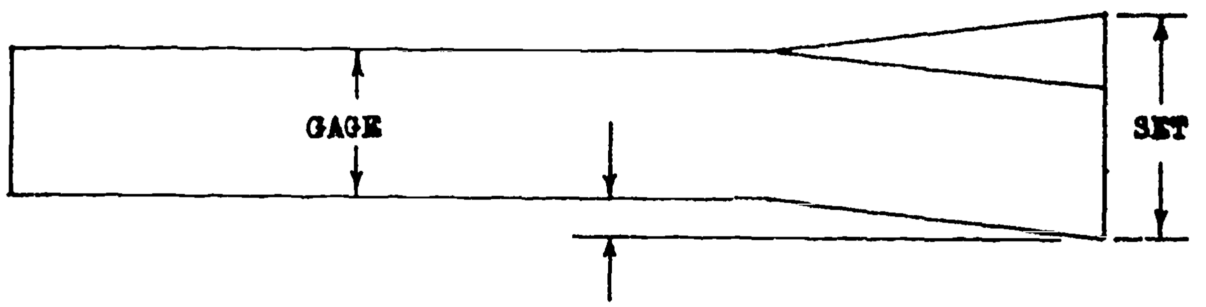
(C) THAT THE MACHINE WILL HANDLE

4. GAGE

ALWAYS USE STANDARD GAGE EXCEPT WHEN INCREASED WORK THICKNESS DECREASES ACCURACY AND WIDTH CANNOT BE INCREASED TO OBTAIN SUFFICIENT STRENGTH. EXAMPLES FOR HEAVY GAGE APPLICATIONS:

(A) WHEN RADIUS CUTTING IN THICK MATERIALS

(B) WHEN MAXIMUM WIDTH USUABLE ON MACHINE PROVIDES INSUFFICIENT BEAM STRENGTH



GAGE IS THE THICKNESS OF THE BAND. SET IS THE AMOUNT THE TEETH ARE OFFSET AND PROVIDES A KERF WIDE ENOUGH TO GIVE CLEARANCE TO THE BAND BACK OF THE TEETH.

BEST COPY AVAILABLE

THE FIVE CHARACTERISTICS THAT MUST BE CONSIDERED EACH TIME THE MACHINIST MUST SELECT A BLADE ARE:

1. TOOTH FORM

(A) FOR FINER THAN 6 PITCH, PRECISION OR REGULAR IS THE ONLY CHOICE.

(B) FOR 6 PITCH AND COARSER, CLAW TOOTH USUALLY GIVES THE BEST TOOL LIFE AND FASTEST CUTTING RATE.

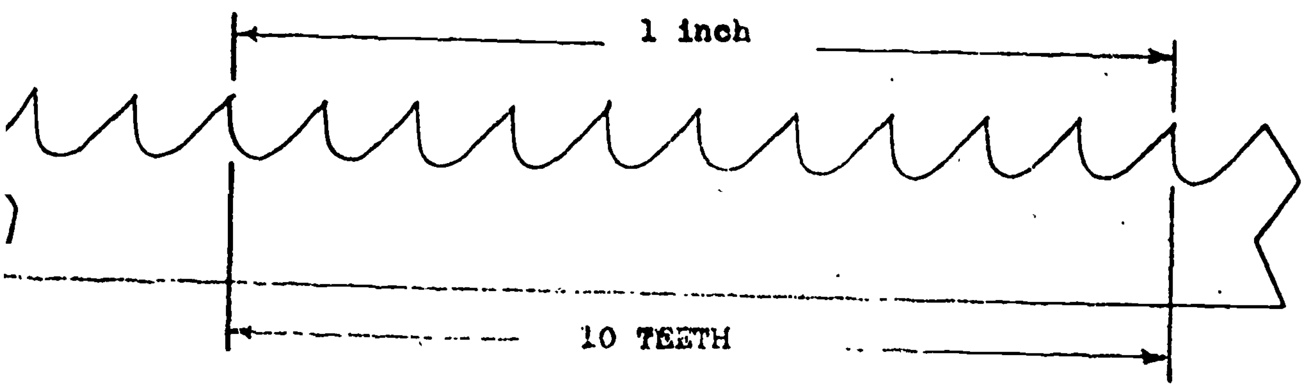
(C) FOR BEST FINISH, PRECISION OR BUTTRESS ARE USUALLY PREFERRED.

2. PITCH - SELECT BEST PITCH FROM SAWING RECOMMENDATIONS TABLE. IF THIS PITCH IS NOT AVAILABLE IN DESIRED WIDTH:

(A) THICK MATERIALS, CHOOSE THENEAREST PITCH.

(B) THIN MATERIALS, REDUCE WIDTH UNTIL PITCH IS FOUND.

(C) HAVE AT LEAST TWO TEETH IN WORK AT ALL TIMES. TEN ARE PREFERABLE FOR HAND FEEDING, 20 FOR POWER FEEDING.



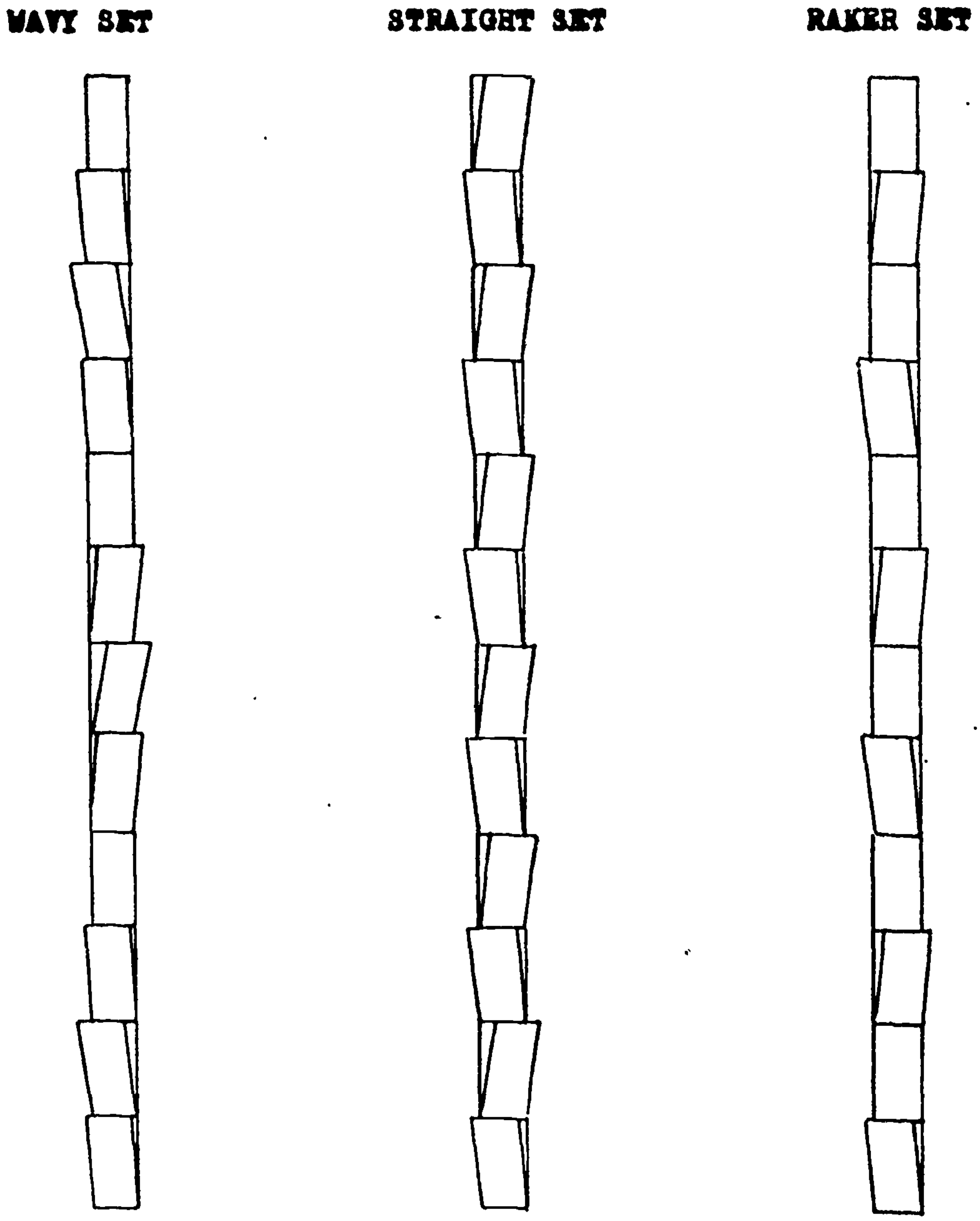
PITCH IS THE NUMBER OF TEETH PER INCH

5. SET PATTERN - ALWAYS USE RAKER SET EXCEPT:

(A) FOR WORK OF VARYING THICKNESS

(B) WHEN ONE BAND MUST BE USED FOR A RANGE OF MATERIAL SIZES - USE WAVE SET

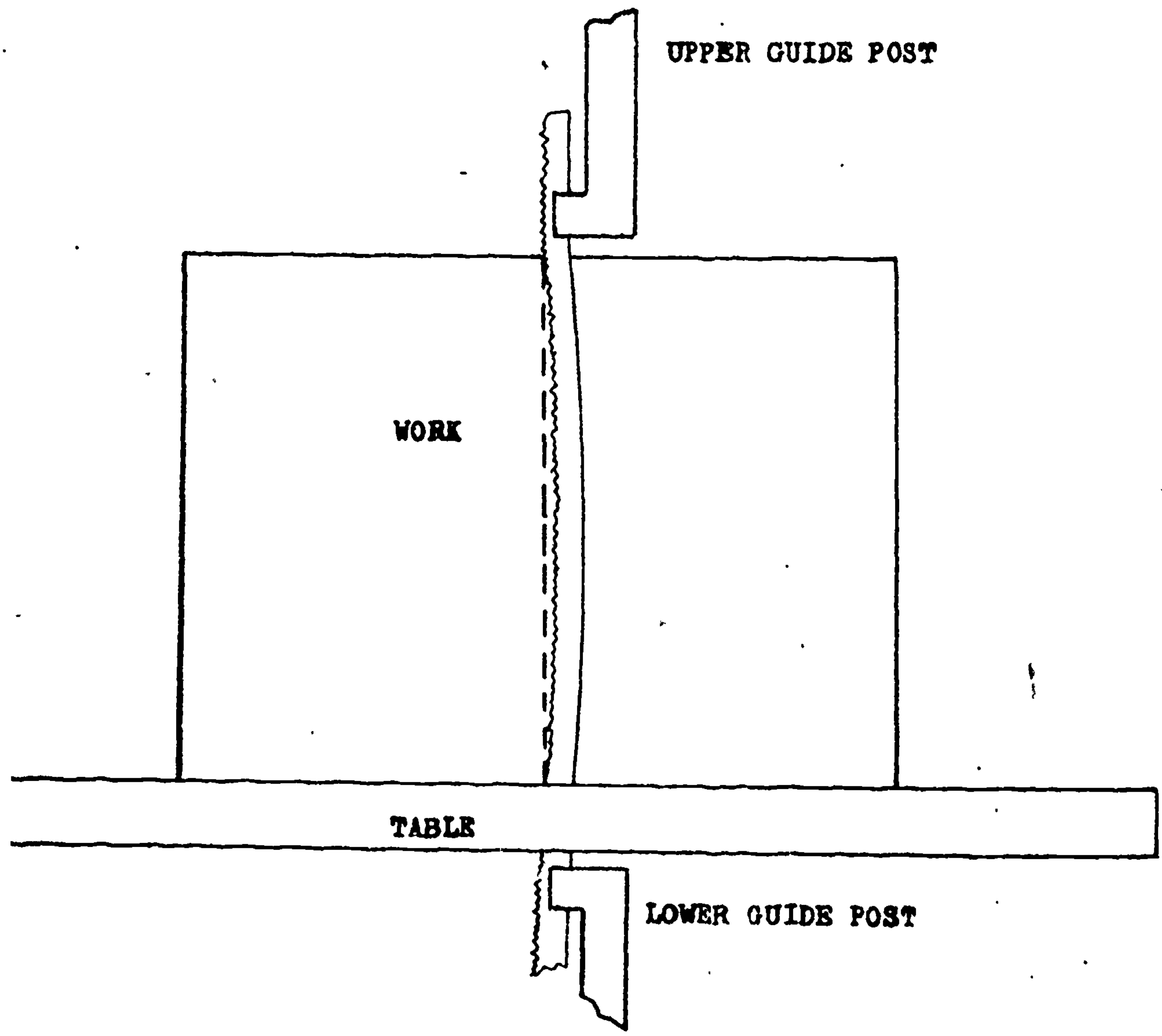
NOTE: THE STRAIGHT SET IS NOT COMMONLY USED IN METALWORKING



THE EFFECTS OF TOO MUCH FEEDING PRESSURE OR OF THE BLADE NOT
ING SUFFICIENT BEAM STRENGTH FOR THE JOB.

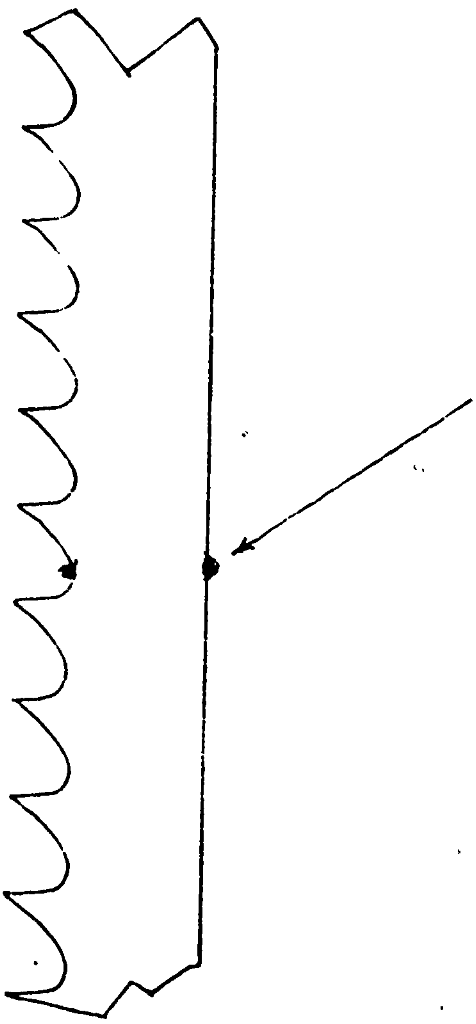
THE CUT WILL NOT BE STRAIGHT NOR AT RIGHT ANGLES. THE DEFLECTION
SES THE BLADE TO HAVE A TENDENCY TO BUCKLE AND TO ''BELLY'' THE
KPIECE, CAUSING AN OUT OF SQUARE CONDITION.

REMEDY: INCREASE WIDTH, GAGE, AND/OR DECREASE FEED.

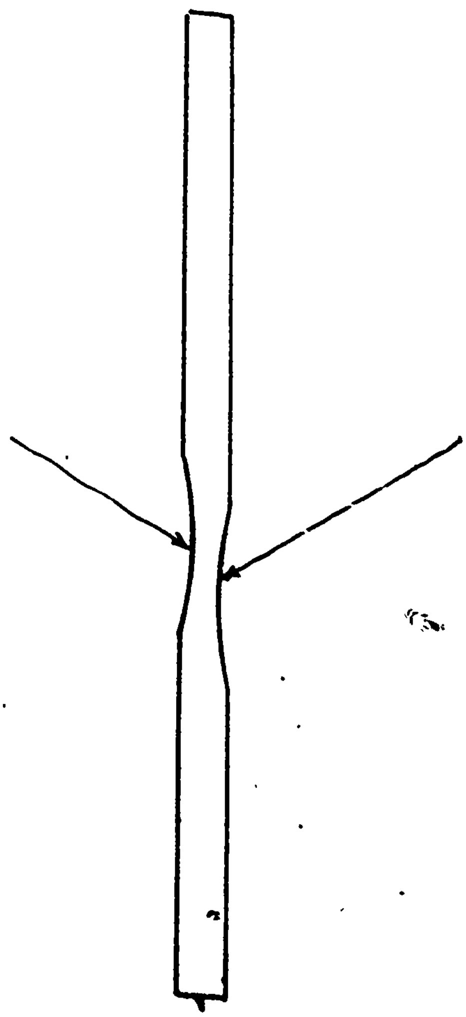


THE TWO MOST COMMON MISTAKES IN MAKING THE WELD

BEAD NOT REMOVED FROM BACK OF BLADE



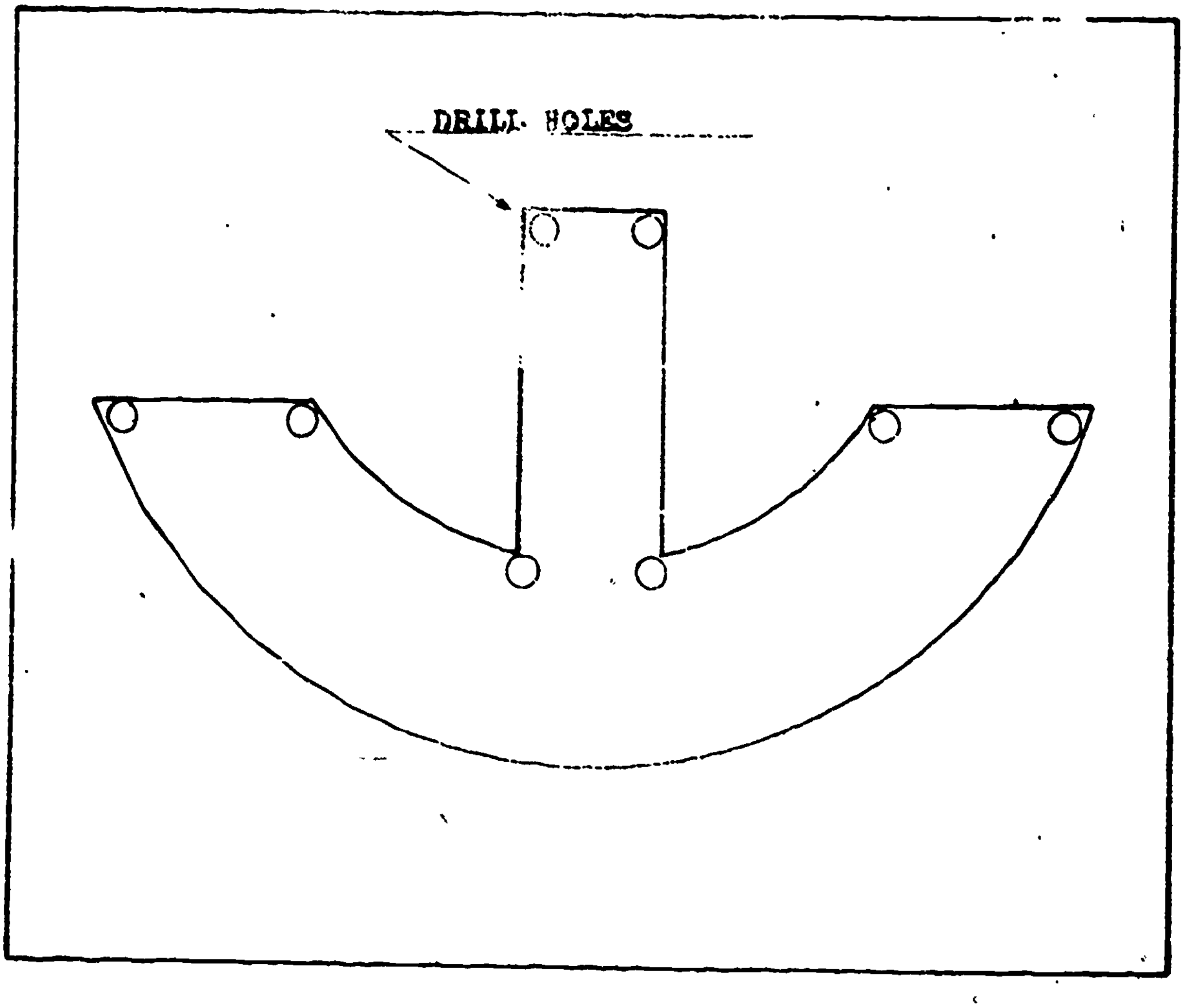
BEAD OVERGROUND - DISHED AND WEAKENED



BEST COPY AVAILABLE

AN EXAMPLE OF DRILLING HOLES IN SCRAP PORTION TO FACILITATE THE SAWING OF INTERNAL CONTOURS.

THE DRILLING OF HOLES ARE ALSO USED FOR EXTERNAL SAWING WHENEVER SHARP CORNERS ARE NOT NEAR THE PERIMETERS. WHEN THE SHARP CORNERS ARE NEAR PERIMETERS, SLOTTING, OR SIMPLY SAWING A KERF TO CORNERS, IS MORE OFTEN USED TO KEEP FROM BACKING SAW OUT OF CUT.



SUMMARIZATION OF PERTINENT FACTORS THAT MUST BE CONSIDERED IN THE SELECTION OF BLADES.

---

1. BLADE CHARACTERISTICS TO MEET JOB REQUIREMENTS:
   (A) TOOTH FORMS - THREE STANDARD FORMS
   (B) WIDTH - FROM 1/16 TO 2 INCHES
   (C) PITCH - FROM 2 TO 32
   (D) GAGE - STANDARD AS WELL AS HEAVY DUTY
   (E) SET - FOR PARTING OR SLOTTING

---

2. RECOMMENDED BLADE FROM WORK PLANNING SYSTEM OR CHART

---

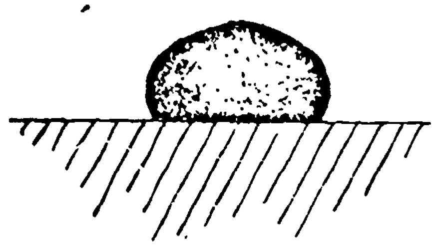
3. PROPER BREAK-IN METHOD, IF NEW BLADE

---

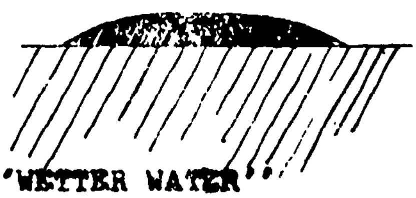
4. OPERATING CONDITIONS FOR LOWEST COST PER CUT
   (A) BAND SPEED - ADJUSTED FOR EACH JOB
   (B) WORK FEED - SELECTED FOR PRODUCTIVITY
   (C) COOLANT - PROPERLY MIXED AND PROPERLY PLACED

---

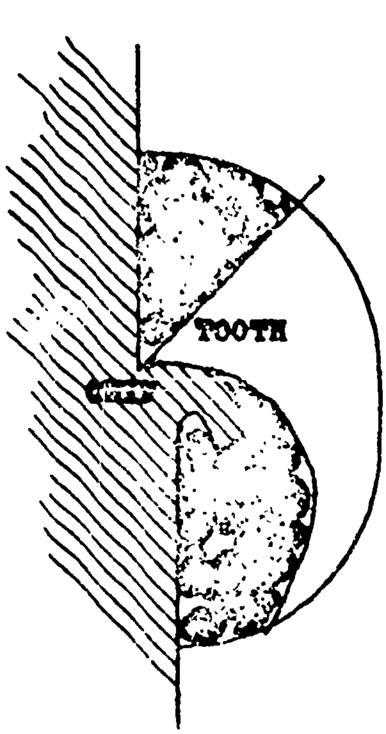
5. BAND REMOVAL POINT FOR MAXIMUM USEFUL LIFE



DUE TO SURFACE TENSION, PLAIN WATER RESISTS SPREADING



''WETTER WATER'' SPREADS OUT



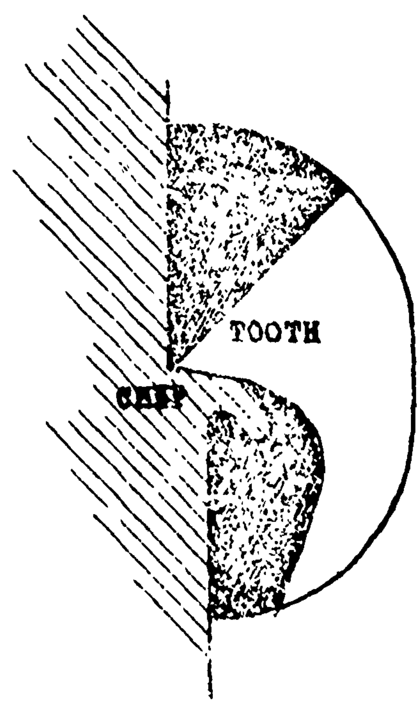
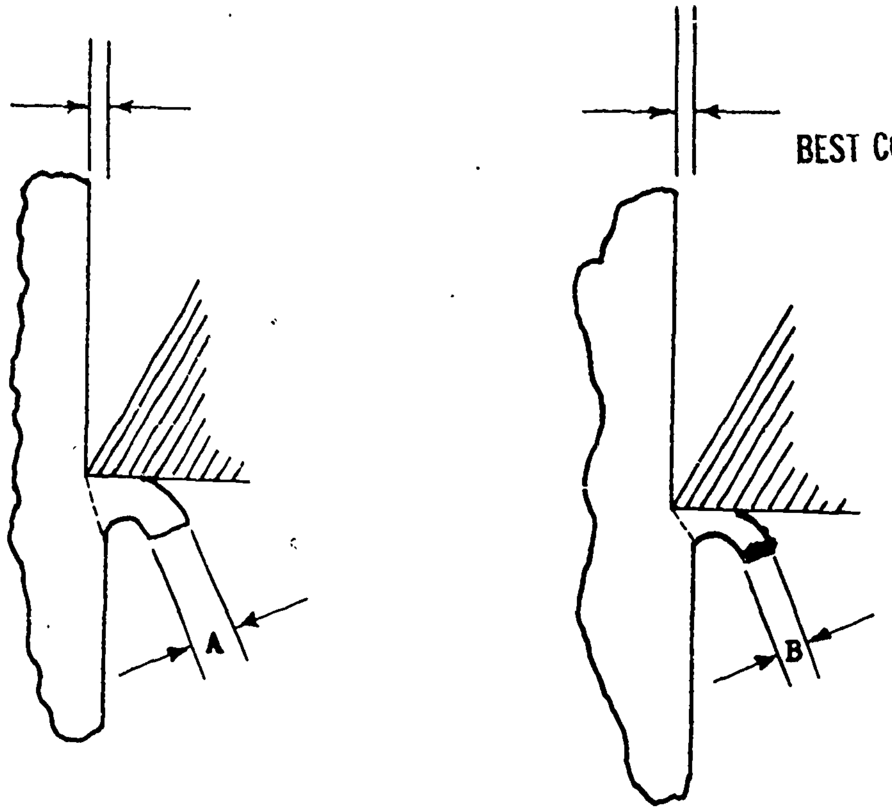
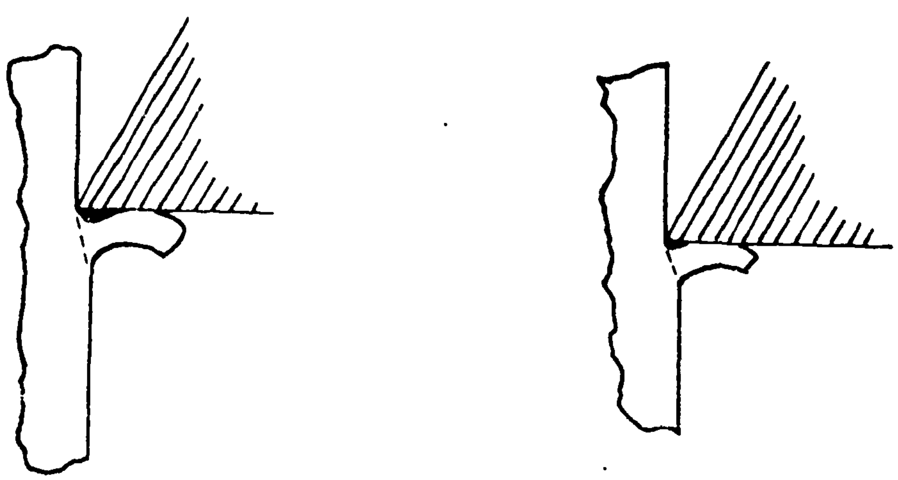


ILLUSTRATION SHOWING HOW PLAIN AND ''WET'' WATER DIFFER IN USE. THE ''WET'' WATER CONTACTS MORE AREA OF TOOTH AND WOULD BE MORE EFFECTIVE AS COOLANT OR AS LUBRICANT.



EXAMPLE ''A'' SHOWS THE LONG SHEAR LINE OF METAL HAVING A HIGH SHEAR STRENGTH, WHILE EXAMPLE ''B'' ILLUSTRATES THE COMPARATIVELY SHORT LINE OF A LOW SHEAR STRENGTH METAL.

NOTE DIFFERENCE BETWEEN WIDTH OF CUT AND WIDTH OF CHIP.



EXAMPLES OF TWO IDENTICAL CUTS BEING MADE ON THE SAME CUT, BUT THE ONE ON LEFT AFTER BUILD-UP OF METAL ON BLADE HAS CAUSED THE SHEAR ANGLE TO INCREASE. THIS CAUSES HEAT INCREASE, A REQUIRED POWER INCREASE, AND SHORTER BLADE LIFE.

AISI
STEEL NUMBERING SYSTEM

The first digit of the 4 or 5 digit numeral designates the type to which the Steel belongs. Thus "1" indicates a carbon steel, "2" a nickel steel, "3" a nickel-chromium steel. The last 2 or 3 digets usually mean carbon content. Thus the symbol 2530 indicates a nickel steel of approx. 5% nickel and .30% carbon content.

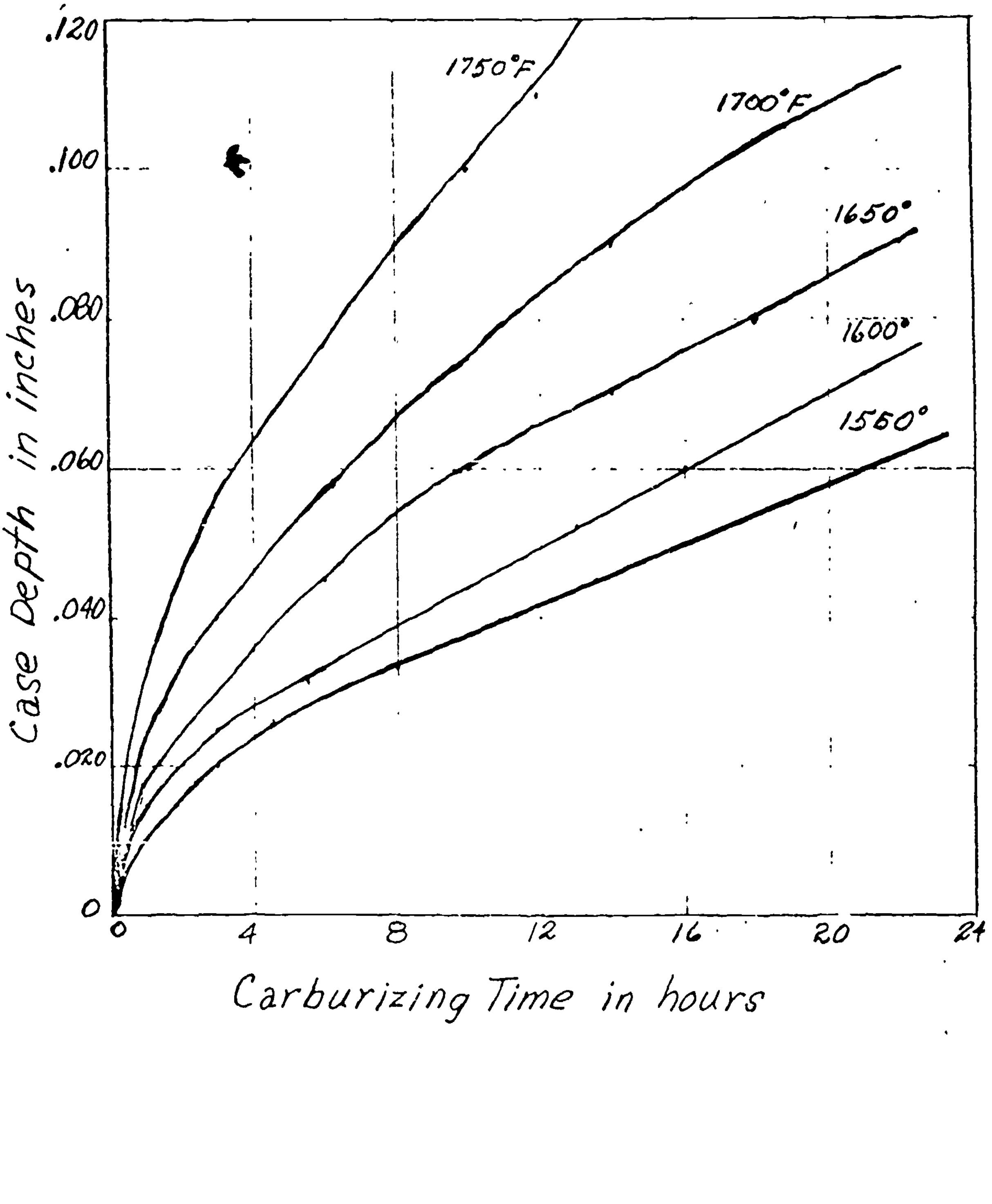
Carbon Steels (basic)	1xxx
Manganese Steels (1.60-1.90%)	13xx
Nickel Steels	2xxx
3.5% nickel	23xx
5.0% nickel	25xx
Nickel-Chromium Steels	3xxx
1.25% nickel- 0.60% chrome	31xx
3.50% nickel- 1.50% chrome	33xx
Molybdenum Steels	40xx
Chrome-Moly steels	41XX
Nickel-chrome-Moly	43xx
Chromium Steels	5xxx
Low chrome	50xx
Chrome-high carbon	52xxx
Chromium-Vanadium Steels	61XX
Silicon-Manganese Steels	92xx

Time Required (per inch) to reach temperature						
Tempering Temperature	hot air oven no circulation			circulating oven air		
	cubes spheres	Squares Cylinder	Flats	cubes Spheres	Squares cylinders	Flats
250° F	30 MIN.	55 MIN	80 MIN.	15 MIN	20 MIN	30 MIN
300°	30	50	75	15	20	30
350°	30	50	70	15	20	30
400°	25	45	65	15	20	30
500°	25	40	60	15	20	30
600°	25	40	55	15	20	30
700°	20	35	50	15	20	30
800°	20	30	45	15	20	30
900°	20	30	40	15	20	30

Soaking time *	Tempering temperature				
1 hour	300° F	325° F	350° F	375° F	400° F
2 hours	280°	305°	330°	355°	375°
4 hours	260°	285°	310°	335°	350°
8 hours	240°	265°	290°	315°	325°

* Soaking time at given temperature - Note that When time doubles, temp. is reduced 20°-25° F. Ref: Tool Steel Simplified pg. 521-2

Case depth – 3115 Steel



Temp. Indicated by Color - Carbon Steel

Degrees C	Degrees F	Color of Steel *
221.1	430	Very pale Yellow
226.7	440	Light Yellow
232.2	450	Pale Straw-yellow
237.8	460	straw-yellow
243.3	470	Deep straw-yellow
248.9	480	Dark yellow
254.4	490	yellow-brown
260.0	500	Brown-yellow
265.6	510	spotted red-brown
271.1	520	Brown-purple
276.7	530	Light purple
282.2	540	Full Purple
287.8	550	Dark Purple
293.3	560	Full Blue
298.9	570	Dark Blue
337.8	640	Light Blue

* Colors indicated will be seen in a room with normal light conditions.

RELATED
INFORMATION

TITLE: Safety in the Shop

UNIT: Safety

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the proper and safe working habits in a machine shop.

REFERENCES:

1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.

2. McCarthy-Smith. Machine Tool Technology. Bloomington, Illinois: McKnight and McKnight Publishing Co.

INTRODUCTION: Safety is aptly described as a thread which is woven throughout the entire fabric of industrial activity. The strength and quality of the fabric are weakened by its absence. Success in industry cannot be achieved by ignoring it. As thread is part and parcel of the cloth, as it assists in tying in one part with the others, so safety is a necessary component of industrial operation, one which cannot afford to be overlooked.

INFORMATION: Safety applied to procedures in a school or industrial shop resolves into using common sense and good judgement. Modern machinery is equipped with guards and other safety devices designed to protect the operator and make operation of equipment as safe as possible. Statistics show that guards and other safety devices afford only 15 percent protection. Most accidents are a result of someone's thoughtlessness, carelessness, lack of knowledge, or lack of consideration for the rights of others and may be avoided by acquiring the habit of thinking before doing.

FOUR POINTS FOR SAFETY

1. The right tool for the job
   Examples of unsafe practices are using a metal chisel with mushroomed end, using files as a pry bar, a wrench for a hammer, and pliers instead of the proper wrench.

2. Tools in good condition

Wrenches with cracked or worn jaws, screw drivers with sharp points or broken handles, hammers with loose heads, dull saws, and extension cords or electric tools with broken plugs or split insulation.

3. Tools used in the right way
   Screwdrivers applied to objects held in the hand, knives pulled toward the body, two hardened steel tools struck together, and failure to ground electrical equipment.

4. Tools kept in safe place
   Many accidents have been caused by tools falling from overhead and by knives, screw-drivers, and other sharp tools carried in pockets or loosely laid in tool boxes.

REASONS FOR UNSAFE PRACTICES

1. Lack of skill.

2. Insufficiently informed or misunderstands.

3. Not convinced of need--indecision.

4. Standard or desired procedure is awkward, embarrassing, uncomfortable, difficult worker prefers nonstandard procedure.

5. Space, light, heat, arrangement, ventilation, materials, tools, equipment, procedure, wages, company policy, or methods or improper, defective, inadequate, inefficient, unsafe, misadjusted, poorly maintained.

6. Physically unsuited, hearing, sight, age, sex, height, weight, disposition, weak, fatigued, ill, nervous, excitable, allergic, or slow reaction.

7. Personal characteristics and attitudes--reckless, bad habits, willfully disobedient, lazy, disloyal, uncooperative, fearful, over-sensitive, jealous, impatient, overambitious, absent-minded, excitable, the 'other fellow' concept (worker beleives himself exempt), inconsiderate or intolerant.

Safety is principally a matter of striving earnestly to learn and follow safe practices and procedures at all times.

RELATED
INFORMATION

TITLE: Finishing

UNIT: Benchwork

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the need and know how to file and polish at the bench.

REFERENCE: 1. Nicholson File Co. File Filosophy.

2. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co.

INTRODUCTION: All machined parts require some bench operation for finishing. These operations may be limited to removing burrs from machinery operations or filling chamfers and polishing.

INFORMATION: The finishing operation may involve filing, polishing, hand lapping and hand scraping. The operation of hand lapping and hand scraping are very slow and skillful operations. The machinist must acquire a skill for these operations.

Filing

Two of the methods used to produce smoothly finished surfaces are draw filing and polishing with an abrasive cloth. Draw filing is a process of filing whereby the file is placed crosswise on the work surface and pulled along. To do draw filing, it is necessary to use a single cut file which will shear rather than clip or scratch the metal. In this operation, great care must be taken to keep the file clean so that scratches will be avoided.

Polishing

Abrasive cloths and papers are used to produce a smooth, polished surface. These cloths and paper come in a great variety of grades from very course to an extremely fine, powdered grade. Some types are used dry while others produce the best results when used wet. There is a grade and type of cloth or paper to suit any purpose, and care should be taken to select the grade which will produce the desired results without wasted time or effect.

Lapping

Lapping is a method of removing very small amounts of material by means of an abrasive. The abrasive material is kept in contact with the sides of a hole that is to be lapped by the use of a lapping tool. The lap should just fit the hole. As the lap revolves in the hole, it should be constantly moved up and down so that the hole will be perfectly cylindrical.

Scraping

Scraping is removing high spots off a flat bearing surface that must be perfectly matched to another flat surface. This is a hand operation requiring much skill.

RELATED
INFORMATION

TITLE: Types of Cold Chisels

UNIT: Bench Work

OCCUPATION: Machinist

OBJECTIVE: To aquaint the student with the four common types of cold chisels.

REFERENCES: 1. Ludwig, O. A., Metal Work Technology and Practice. Bloomington, Ill. McKnight and McKnight Publishing Co.

2. Nicholson, Fred, Shop Theory. New York: McGraw-Hill Book Co., Inc.

INTRODUCTION: Cold chisels are used to cut cold metal, hence the name. They are made of high carbon tool steel in a number of sizes and shapes. The cutting end of a chisel is hardened and temptered because the cutting edge must be harder than the metal it is to cut. Chisels are usually made from octagon shaped steel 3/8" to 1" in diameter and from 6" to 12" long.

INFORMATION: Types of Chisels: The types of cold chisels are designated by the shape of their cutting edges. The following are the four types of chisels and some of their uses:

a. Flat Cold Chisel: The flat chisel (Fig.1) has a wide cutting edge. It is used for chipping flat surfaces, cutting sheet metal, bars, rivets, bolts and for most of the ordinary chipping around the shop. The cutting edge is sharpened to 70 angle. The flat chisel is the most common type of cold chisel.

Fig. 1. Flat Cold Chisel

b. Cape Chisel: The cape chisel (Fig.2) has a narrow cutting edge and is used mostly for cutting narrow grooves and key ways. It is widest at the cutting edge to keep the chisel from sticking in the grooves and maybe breaking.

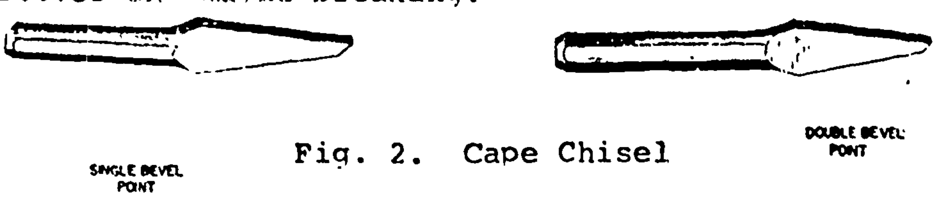


Fig. 2. Cape Chisel

c. Round Nose Chisel: The round nose chisel (Fig.3) has a rounded cutting edge and is used for cutting u-shaped grooves and chipping filleted corners.

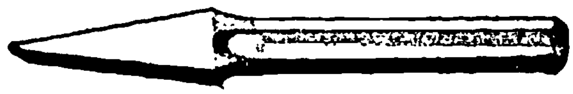


Fig. 3. Round Nose Chisel

d. Diamond Point Chisel: The diamond point chisel (Fig.4) has a cutting edge shaped like a diamond and is used to, for cutting v-shaped grooves and for chipping in sharp corners.

Fig. 4. Diamond Point Chisel

RELATED
INFORMATION

TITLE: To Read a Micrometer Caliper

UNIT: Bench Work

OBJECTIVE: To give the student practice in reading the micrometer caliper.

INTRODUCTION: Every machinist must be able to read a micrometer if he is to do accurate work. A regular micrometer will read accurately to .001 of an inch.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc. Chapter 6 Fig 3-64 and 3-66.

INFORMATION: Procedure:

1. Open the micrometer until it will pass over the work.

2. Set the anvil of the micrometer on one side of the work, holding the frame of the micrometer lightly in the hand.

   NOTE: Never attempt to measure moving work with the micrometer calipers.

3. Screw the thimble of the micrometer down until the end of the spindle will just slide over the work. Keep the contact points in a line at a right angle to the work.

4. Remove the micrometer from the work being careful not to move the thimble.

5. Note the last number showing on the barrel ahead of the thimble. This figure represents the number of tenths of on inch that the micrometer is open, because each division on the barrel is equal to .025". Every fourth line is numbered and is equal to .100". In Fig. 3-66 page 72 in Shop Theory this is number 2 and is .200".

6. Note how many divisions marks are visible past the last numbered line. Each of these represents .025". In Fig. 3-66, one division is showing, and 1x .025 = .025.

7. Note the number from 0 to 25, of mark on the thimble which coincides with the revolution line on the barrel. Each of those lines represents

.001". In Fig. 3-66, this is number 16 and is .016".

8. Add the results in steps 5, 6, and 7. The results is .200 + .025 + .016 = .241.

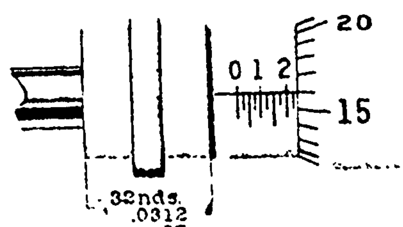
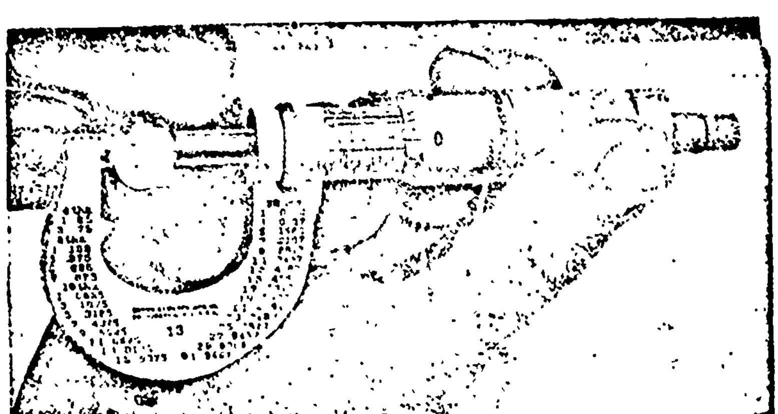


FIG. 3•66. A reading of 0.241 in. on a micrometer. (Brown & Sharpe Mfg. Co.)



How to hold the micrometer correctly when piece is held in the hand. Note positions of the fingers. (*The Brown & Sharpe Manufacturing Company*)

RELATED
INFORMATION

TITLE: To Read An Inside Micrometer and A Depth Micrometer.

UNIT: Bench Work

OCCUPATION: Machinist

OBJECTIVE: Every machinist must be able to read a depth micrometer if he is to do accurate work. A depth micrometer will read accurately to .001 of an inch.

REFERENCE: Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.

INTRODUCTION: The reading of an inside micrometer and a depth micrometer are the same, so the following applies to both types of micrometers. The explanation of how to read will be given only for that of a depth micrometer.

INFORMATION: Each division on the line or barrel represents .025 of an inch; every fourth line is numbered as on a plain micrometer and these numbers represent hundred thousands or .500, .600, .700, etc. Thimble .001/in.

Each line on the sleeve or thimble represents one thousands of an inch or .001, .002, .003, etc.

NOTE: When reading the depth micrometer that the same general rule applies as used on a plain micrometer only that the scale on the barrel and sleeve are laid out just the opposite of those on a plain micrometer. Those on the barrel increase in value from right to left instead of from left to right as a plain micrometer, and those on the sleeve increase in value counterclockwise instead of clockwise as on a plain micrometer.

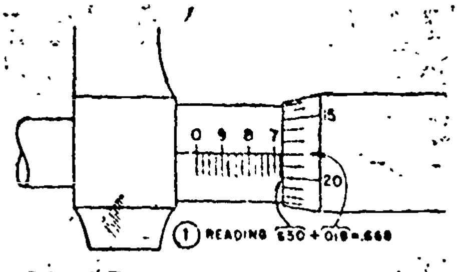
Because of the difference in scales you must remember to use the value of the next hidden line on the barrel as your barrel reading. In other words, the last line covered by the end of the sleeve. To find what the value of the hidden barrel line is simply check to see what the value of the lines are which can be seen on the barrel. In the example shown note the last numbered line which can be seen, closest to the end of the sleeve is the numbered 7 line, so you know what your barrel reading is going to be .600 plus.

Now notice that one line on the barrel is visible between the numbered 7 line on the barrel and the end of the sleeve. Each line on the barrel represents .025 of an inch. Now add this .050 to your hidden numbered line which is .600. Your total barrel reading is then .600 + .050=.650.

To obtain your sleeve or thimble reading, simply take the value of the sleeve line which matches the horizontal index line on the barrel. In this case the number is 18 line is the matching line so add .018 to your barrel reading for a total reading of .650 + .018 = .668.

If extension rods are used as in often the case, remember to add the length of the rod to your measurement reading.

EXAMPLE: If in the reading shown, a 3 inch-long rod was being used you would add only two inches to the measurement of .668 and your total reading would be 2.668. The rod is 3 inches long but only two inches enter into the measurement, as 1 inch of the rod length is used in fastening the rod into the head of the micrometer.



RELATED
INFORMATION

TITLE: To read a Vernier Caliper

UNIT: Bench Work

OCCUPATION: Machinist

OBJECTIVE: To give the student practice in reading the vernier caliper.

REFERENCE: McCarthy-Smith. Machine Tool Technology. Bloomington, Illinois. McKnight and McKnight Publishing Co.

INTRODUCTION: Every Machinist must be able to read a vernier caliper if he is to do accurate work. He needs to read a vernier caliper for work larger than the micrometer he has in his tool box and the vernier height gage for layout.

INFORMATION: The top or main scale divisions are indicated by three different length lines. The longest vertical lines on the main scale are numbered with larger numbers, and they represent inches. The medium length lines are numbered with smaller numbers from 1 to 9 and represent hundreds of thousands--.100, .200, .300, etc. The shortest vertical lines on the main scale each represent twenty-five thousands or .025.

NOTE: To read the top, or main scale, keep in mind that the zero line on the lower vernier scale determines your reading of the top, or main scale.

First notice the value of the whole inch line to the left of the zero line on lower scale.

EXAMPLE: The number 6 line of the top scale is the first inch line to the left of zero line on lower scale so our whole inch reading is 6.000.

Next we count the number of lines showing between the six inch of the top scale and the zero line of the lower scale.

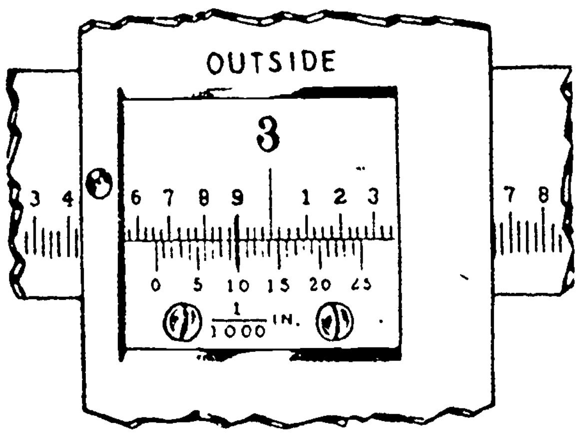
EXAMPLE: There are two lines so add 2x.025 or .050 to the whole inch reading of 6.000. 6.000 + .050 = 5.050.

To read the lower vernier scale remember that each line on this scale equals .001, or one thousands of an inch. To obtain this reading, simply take the value of the vernier scale line which matches a line on the top scale.

EXAMPLE: The number 10 line is the matching line so add .010 to the top scale reading. 6.050 + .010 = 6.060.

If the zero line on the vernier scale matches a line on the top of the scale the vernier scale reading will be zero. In this case you simply take the value of the line on top scale which matches the zero line on the lower scale.

EXAMPLE: If the reading shown, the zero line on lower scale matches the number 1 line on top scale, the total reading would be 6.000 + .100 = 6.100.



RELATED
INFORMATION

TITLE: Drills and Drill size

UNIT: Drills and Drilling Processes

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the process of drills and drill sizes.

REFERENCES: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. McCarthy-Smith. Machine Tool Technology. Bloomington, Illinois: McKnight and McKnight Publishing Co.

INTRODUCTION: Early drills were made of flat pieces of steel twisted to form the helical flutes. After the forging and rough machining operations the drill was hardened and ground and ready then for use.

INFORMATION: Drills today are made of high speed alloy steels which will permit operations at much higher speeds without effect on the hardness of the drill as compared to the earlier drills. The material used for drills today is cylindrical. The flutes and clearances are machined on special production type machines. The hardening process is controlled to insure uniform hardness. The sharpening is done on special machines designed to provide maximum efficiency and a longer life for the drill.

The majority of holes that are drilled are smaller than 1/2" in diameter. To meet this need the manufacturers have produced a number of standard drill sizes smaller than 1/2" in diameter. Fractional size drills are available from 1/64" to 1/2" and larger in diameter in multiples of 1/64". Number drills, from #1 to #80, provide eighty different drill sizes to supplement fractional sizes smaller than 1/4" in diameter. Letter size drills from A to Z supplement the fractional sizes from 1/4" to 1/2" in diameter. These three standard drill sets provide over one humdred and thirty different diameters which may be used by the machinist today.

Straight shank drills with carbide tips are recommended for high-production drilling of cast iron, cast steel, and nonferrous materials. They are not recommended for drilling steel. Carbide die drills of the straight-shank type are recommended for use in drilling hardened steel in the range from 48 to 65 Rockwell-C hardness. Holes may be drilled without annealing the metal. A steady hand feed with a good flow of cutting fluid should be used at a cutting speed from 75 to 100 RPM.

BEST COPY AVAILABLE

RELATED
INFORMATION

TITLE: Speeds, Feeds for Drilling

UNIT: Drills and Drilling Processes

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the proper speeds and feeds for different types of materials.

REFERENCES:

1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. Burghart-Axelrod-Anderson. Machine Tool Operation Part I. New York: McGraw-Hill Book Co., Inc.
3. Porter-Lascoe-Nelson. Machine Shop Operations and Setups. Chicago, Illinois: American Technical Society.

INTRODUCTION: The feed of the drill bit into the metal varies according to the hardness or softness of the material being drilled. Too much feed can cause a damaged drill or work piece. On a power operated down feed drill press the feed can be set to a certain depth per revolution. Charts in text books with recommended speed and feeds are available. The feed is expressed in surface feet per minute. The speed is expressed in revolution per minute.

INFORMATION:

SPEEDS FOR DRILLING

The speed of a drill is usually measured in terms of the rate at which the outside or periphery of the tool moves in relation to the work being drilled. The common term for this is "Surface Feet Per Minute", abbreviated to S.F.M. The relation of S.F.M. and Revolutions Per Minute, or R.P.M. is indicated by the following formulas:

S.F.M. = .26 X R.P.M. X Drill Diam. in Inches

$$R.P.M. = \frac{4X \text{ Cutting Speed}}{\text{Drill Diameter}}$$

In general, when operating a drill at a speed anywhere within its range for the particular material involved, increases in speed result in fewer holes before grinding becomes necessary and reductions in speed permit more holes before the tool is dulled. As a result, on every job there is the problem of choosing a speed which will permit the highest rate of production without entailing excessive drill costs or down-time for tool sharpening. The most efficient speed for operating a drill will depend on many variables, some of which are listed below:

1. Composition and hardness of material.
2. Depth of hole.
3. Efficiency of cutting fluid.
4. Type and condition of drilling machine.
5. Quality of holes desired.
6. Difficulty of set-up.

Speeds shown in the following table indicate the approximate range for efficient operation under normal conditions. On most jobs, adjustments from these speeds will be required to reach peak efficiency.

MATERIAL	SPEED IN S. F. M.
Alloy Steel 300 to 400 Brinnel	20-30
Stainless Steel	30-40
Tool Steel, 1.2 Carbon	50-60
Steel .4 to .5 Carbon	70-80
Mild Machinery Steel .2 to .3 Carbon	80-110
Hard Chilled Cast Iron	30-40
Medium Hard Cast Iron	70-100
Soft Cast Iron	100-150
Malleable Iron	80-90
Monel Metal	40-50
High Tensile Strength Bronze	70-150
Ordinary Brass and Bronze	200-300
Aluminum and its alloys	200-300
Magnesium and its alloys	250-400
Slate, Marble, and Stone	15-25
Bakelite and similar material	100-150
Wood	300-400

* The above table is for the use of High Speed Drills - if Carbon Steel Drills are used the speed should be 40 to 50 percent of the speed listed.

FEEDS FOR DRILLING
The feed of a drill is governed by the size of the tool and the material drilled. Since the feed partially determines the rate of production and also is a factor in tool life, it should be chosen carefully for each particular job. In general the most effective feeds will be found in the following range:

DRILL DIAMETER IN INCHES	FEED IN INCHES PER REVOLUTION
Under 1/8	.001 to .002
1/8 to 1/4	.002 to .004
1/4 to 1/2	.004 to .007
1/2 to 1	.007 to .015
1 and over	.015 to .025

RELATED
INFORMATION

TITLE: Twist Drill Failures

UNIT: Drills and Drilling Processes

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with twist drill failures and their causes.

REFERENCES:
1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. Burghart-Axelrod. Machine Tool Operation. Part 1. New York: McGraw-Hill Book Co., Inc.
3. Porter-Lascoe-Nelson. Machine Shop Operations and Setups. Chicago, Ill: American Technical Society.

INFORMATION: Twist Drill Failures and Their Causes

1. Drill Breakage:
   Caused by
   - spring or backlash in press or work
   - too little lip clearance
   - too slow speed in proportion to the feed
   - dull drill
   - improper chip clearing by drill (clogging).

2. Broken Tang:
   Caused by
   - imperfect fit of taper shank in the socket, due to nicks, dirt, burrs, or worn-out socket.

3. Breaking down of outer corners of cutting edge or lip:
   Caused by
   - too much speed
   - too much lip clearance
   - improper cutting compound
   - no lubricant at point of drill
   - improper chip clearing by drill
   - material being drilled has hard spots, scale or sand inclusions.

4. Oversize Hole:
   Caused by
   - unequal angle of point
   - unequal length of cutting edge
   - loose spindle
   - wrong drill (check size before using the last fellow may have put it in the wrong location.)

BEST COPY AVAILABLE

5. Rough Hole:
   Caused by - dull drill
   - improperly ground drill
   - wrong lubricant, or lack of lubricant
   - too much pressure
   - too much feed.

6. Splitting up center:
   Caused by - too little lip clearance
   - too much feed.

7. Checks or cracks in cutting edge:
   Caused by - overheated drill
   - too quickly cooled while sharpening or drilling.

8. Margin chips:
   Caused by - oversize jig bushing
   - mishandling in storage or use.

9. Drill will not enter work:
   Caused by - dull drill
   - too small lip clearance
   - web too heavy
   - material excessively hard
   - chip caught between drill point and work

A FEW "DON'T" FOR DRILL PRESS OPERATIONS

1. Don't change belt with motor running.
2. Don't try to hold work by hand--get a clamp or vise.
3. Don't force the drill -- you will dull or break it.
4. Don't try to stop revolving work -- a broken drill is cheaper than a broken finger or hand.
5. Don't take chances -- if you are not sure, ask your superior.
6. Don't "work harden" the drilled hole by using too fine a feed or a dull drill--subsequent use of a tap or reamer will result in damage or breakage.

RELATED
INFORMATION

BEST COPY AVAILABLE

TITLE: Speeds, Feeds for Reaming

UNIT: Reaming Processes

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the proper speed, feed and amount of stock removal with machine reamers.

REFERENCE: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. McCarthy-Smith. Machine Tool Technology. Bloomington, Illinois: McKnight and McKnight Publishing Co.

INTRODUCTION: Reaming is used to obtain accurate sized holes with a smooth finish. Accuracy and finish can be obtained by using several drills in successive order and stoning the cutting edge of the final drill. This method is time consuming, and care must be used to obtain the same results as reaming. Reamers which are power driven are called machine reamers. Machine reaming is fast and sufficiently accurate for most of the work done in a machine shop.

INFORMATION: SPEEDS
The most efficient speed for machine reaming is closely tied in with the types of material being reamed, the rigidy of the set-up, and the tolerance of the finish required. Quite often the best speed is found to lie around two-thirds of the speed used for drilling the same material.
A lack of rigidity in the set-up may necessitate slower speeds, while occasionally a very compact, rigid operation may permit still higher speeds. When close tolerances and a fine finish are required it is usually found necessary to finish ream at considerably slower speeds.

FEEDS
In reaming, feeds are usually higher than those used for drilling. The amount of feed may vary with the material but a good starting point would be between .0015" and .004" per flute per revolution. Too low a feed may result in glazing, excessive wear, and, occasionally, chatter. Too high a feed tends to reduce the accuracy of the hole and may also lower the quality of the finish. The basic idea is to use as high a feed as possible and still produce the required finish and accuracy.

AMOUNT OF STOCK REMOVED
For similar reasons, insufficient stock for reaming may result in a burnishing rather than a cutting action. It is difficult to generalize on this phase, as it again is closely tied to in with type of material, feed, finish required, depth of hole and chip capacity of the reamers. For machine reaming, .010" a 1/4" hole, .015" on a 1/2" hole, up to .032" on 1 1/2" hole, seems a good starting point.

RELATED
INFORMATION

TITLE: Single Point Cutting Tools

UNIT: Lathe Work

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the correct angle, shape of the tool, smoothness and the selective of the correct type of tool for the material to be machined.

REFERENCE: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. How to Run a Lathe. South Bend Lathe Work.

INTRODUCTION: The cutting efficiency of the single point cutting tool is the responsibility of the machinist. The cutting tool efficiency is judged by the tools ability to remove material, the quality of the finish, the amount of machining achieved by the cutting tool before regrinding becomes necessary.

INFORMATION: The grinding of a lathe cutting tool involves consideration for two types of angles: clearance angles and rake or cutting angles. As the name implies. clearance angles are provided so that the cutting edge can be brought in contrast with the work surface without interference from the body of the tool. Clearance angles are ground on the front of the tool and on the side of the tool which is being fed into the work. These angles do not affect the cutting action of the tool. Their sole function is to provide the necessary clearance to the cutting edge without sacrificing strength.

The rake angle on a lathe tool determines how the tool will cut and the shape and direction of the chip. These angles vary with the material being cut. For turning brass we use a tool that has no side rake, since side rake would cause the tool to "dig in" and tear the material. A tool for turning steel must have considerable side rake to reduce the tremendous cutting pressure, caused by this harder, tougher material.

Tables of recommended clearance and rake angles for machining the various materials are listed in most text books and handbooks. In the actual grinding operation the new machinist student will find it necessary, at first, to use a protractor or a tool gage to check accuracy. With repeated practice, these aids can be eliminated.

BEST COPY AVAILABLE

RELATED
INFORMATION

BEST COPY AVAILABLE

TITLE: Methods of Holding Work in on a Lathe

UNIT: Lathe

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the different methods of holding work in the lathe while being machined, proper speeds, and feeds to use while operating the lathe.

REFERENCE: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. How To Run a Lathe. South Bend Lathe Work.

INTRODUCTION: The machinist needs to know the different methods of holding work in the lathe and how to calculate his speed and feeds.

INFORMATION: Any machining job involves five steps before completing. These steps, in general, are the same for all machines:

1. Mounting or holding the work while it is being machined.
   a. Between centers. The work is supported at each end by concal points called centers.
   b. In a chuck. One end of the work is held by the jaws of the chuck. The other end is exposed for machining. There are two types of chucks - three jaw combination which is self-centering and four jaw type in which each jaw is independent of the others.
   c. Face Plate. The work is bolted to this device.
   d. Collets. Self-centering holding devices used primarily in production work.

2. Selecting and Setting Proper Speed.
   Use formula to determine the proper Speed:
   $$R.P.M. = \frac{4\,(C.S.)}{D}$$
   C.S. = Cutting Speed
   D. = Diameter of work
   Available speeds are usually given on the machines. These are listed inside the head- [illegible] es or on the headstock cover.

3. Selecting and Setting proper Feeds.
   The following feeds are recommended on lathes: .015" per revolution for roughing cuts;

and .0038" per revolution for finishing feed.

4. Selecting and Mounting Cutters
   A cutter is selected for a specific operation or cut to be made. For lathe work these are called:
   1. Round nose turning tools
   2. Facing Tools
   3. Threading Tools

   Lathe tools or cutters should be mounted "on center" and roughly perpendicular to the face being machined. A cutter is "on center" when the top or cutting edge is level with the centers of the work.

5. Measuring.
   This is done both on the machine and with hand measuring devices found in the tool trays. When the micrometer is used, measurement is in thousandths of an inch. The measuring devices on the machines, except for the drill press, read in thousandths also.

RELATED
INFORMATION

TITLE: Boring on the Lathe

UNIT: Lathe Work

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the boring operation on the lathe.

REFERENCES: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. How To Run a Lathe. South Bend Lathe Work.

INTRODUCTION: Boring is an internal turning operation. It is possible to turn a diameter, cut a thread, taper or chamfer, or perform any operation internally that can be performed externally on the work piece.

INFORMATION: The principle of stock removal does not change whether it is external or internal operation on the work piece. The movement of the crossfeed screw is reversed for boring operations, the tool bit must be held in a special tool holder that will permit it to be fed deeply into a hole. The tool bit must have more front clearance to avoid rubbing.

Holes which are to be bored are usually roughed out with drill held in the tailstock. The boring tool is relatively springy, therefore the hole is drilled as large as possible, with a core drill, leaving a minimum of stock for boring. On castings holes are cored that are to be bored to save time in the machining.

Boring requires a greater skill on the part of the machinist than does external turning. He must be able to tell by the sound of the cut, the condition of the chip and the condition of the tool how the operation is progressing.

RELATED INFORMATION

TITLE: Types of Tapers

UNIT: Lathe Work

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the more common tapers used in machine tool work.

REFERENCE:

1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. Axelrod, Aaron, Machine Shop Mathematics. New York: McGraw-Hill Book Co., Inc.

INTRODUCTION: Taper is the difference in diameter for a unit; of length of a conical piece of work. It might also be considered as a uniform rate of change in the diameters of a conical piece of work. Several machines in the machine shop that have revolving spindles which have tapered holes into which the tapered shanks of drills, reamers, centers, and so forth, are fitted and securely held in place by the holding action of mating tapers.

INFORMATION: TYPES OF TAPERS: The following are the more common types of tapers used in the machine shop.

MORSE: The Morse taper is the most common at the standard machine tapers. It is used on most taper-shanked drills, reamers, and other taper shanked tools used on large drill presses and lathes. There are eight sizes of morse tapers, numbered from 0 to 7, 0 being the smallest and 7 the largest. The morse taper is approximately 5/8" per foot.

BROWN AND SHARP: The Brown and Sharp is a standard form of taper that is used mostly on old style milling machines and tapered shanked and mills. There are 18 sizes of the Brown and Sharp tapers, numbered from 1 to 18, with 1 being the smallest and 18 being the largest. The amount of taper is 1/2" per foot.

JARNO: Jarno taper is standard form of taper that may be found on some special purpose machines. There are 20 sizes of the Jarno taper ranging from 1 to 20, with 1 being the smallest and 20 the largest. The amount of taper is .600" per foot.

BEST COPY AVAILABLE

STANDARD MILLING MACHINE: The standard milling machine taper is used in the spindle of most late model milling machines and on the arbors and adaptors for these machines. There are four standard sizes of this taper numbered 30 the smallest, 40, 50, and 60 the largest. The amount of taper is 3 1/2" per foot.

STANDARD TAPER PIN: Taper pins are used to align or hold parts together. They are preferred to straight pins where they must be removed from time to time. There are 14 sizes, numbered from 0 the smallest to 13 the largest. The amount of taper is 1/4" per foot.

TAPER FORMULAS AND SYMBOLS: The amount of taper is usually specified either by the taper per foot, by the length of taper and the diameters at the ends of the taper, or by degrees. The following formulas may be used to make most taper caculations.

RELATED INFORMATION

TITLE: Taper Formulas

UNIT: Lathe Work

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the symbols and formulas used when making taper calculations.

REFERENCE: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. Axelrod, Aron. Machine Shop Mathematics. New York: McGraw-Hill Book Co., Inc.

INTRODUCTION: The student must know and understand the symbols and formulas for taper calculations. He must be able to make calculations for the tailstock off-set method and the taper attachment.

INFORMATION: SYMBOLS USED:

Lt = Length of taper in inches
Ls = Length of entire workpiece
D = Large diameter
d = Small diameter
TPI= Taper per inch
Offset = Tailstock set over in inches
TPF= Taper per foot

FORMULAS:

$$TPI = \frac{D-d}{Lt}$$

$$TPF = \frac{D-d}{Lt} \times 12$$

$$D = \frac{TPF}{12} \times Lt + d$$

$$d = \frac{TPF}{12} \times Lt - D$$

$$Lt = \frac{D-d}{TPF} \times 12$$

$$Offset = \frac{Le}{12} \times \frac{TPF}{2}$$

$$Offset = \frac{Le}{Lt} \times \frac{D-d}{2}$$

RELATED
INFORMATION

TITLE: Mechanical Accessories--Fasteners

UNIT: Screw Threads

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the different types of fasteners or holding devices used in the machine shop.

REFERENCE:

INTRODUCTION: Fasteners may be classified as either permanent fasteners like rivets, or movable fasteners, such as bolts, screws, and keys. Movable fasteners are most widely used in a machine shop.

INFORMATION: Bolts--Hexagon-head or square- head bolts for general fastening purposes. The length of bolts is always given from under the head to the point, and the length of threaded portion is given from the point.

Setscrews--The purpose of setscrews is to prevent motion between two parts, such as a pulley on a shaft. Setscrews are made with either headless or with square heads and with many point shapes.

Capscrews--There are six standard heads for cap-screws, namely: hexagon head, flathead, button head, fillister head, hexagonal socket head, and fluted socket.

Machine Screws--Machine screws differ from cap screws, chiefly in that they are smaller. Machine screws are adapted for use with materials of thin section.

Keys--Keys are used to transmit positive motion between shafts and pulleys, cranks, etc.

Taper Pins--Taper pins are used in fastening hubs, collars, hand wheels, etc., to shafts and generally for parts which must be separated frequently.

Dowel Pins--Dowel pins are used to maintain initial alignment between parts and to prevent any side-wise movement between parts.

BEST COPY AVAILABLE

RELATED
INFORMATION

TITLE:	Screw Thread Terms
UNIT:	Screw Threads
OCCUPATION:	Machinist
OBJECTIVE:	To acquaint the student with the terms that apply to the American National Screw Thread form.
REFERENCES:	Anderson-Tatro. Shop Theory. New York; McGraw-Hill Book. Co., Inc. Burghard, H. D. Axelrod, Aaron and Anderson, James. Machine Tool Operation Part 1. New York: McGraw-Hill Book Co., Inc.
INTRODUCTION:	The American National Screw Thread form is the most common screw thread form used in this country. The three series of this thread are the National Course (NC). The difference being the number of threads per inch. The following terms relate to the American National Screw Thread form and most other screw tharead forms as well.

TERMS:

Screw Thread--A ridge of uniform section in the form of a helix on the external or internal surface of a cylinder or cone.

External Thread--Thread on the outside of a member.

Internal Thread--Thread on the inside of a member.

Major Diameter--The largest diameter of a screw thread.

Pitch Diameter--The diameter of an imaginary cylinder or cone that would cut through the threads at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder.

Crest--The top surface joining the two sides of a thread.

Root--The bottom of the surface joining the sides of a thread.

Side--The surface of a thread which connects the crest with the root.

Depth of Thread--The distance between the crest and base of the thread.

Length of engagement--The length of contact between two mating parts, measured axially.

Fit--The relationship of two mating parts with reference to ease of assembly. Four classes of fit have been established by the National Screw Thread Commission for the purpose of insuring the interchangeable manufacture of screw threads in this country, the number and corresponding fits are as follow:

No. 1	Loose fit	No. 3	Medium fit
No. 2	Free fit	No. 4	Close fit

Minor Diameter--The smallest diameter of a screw thread.

Pitch--The distance from a point on a screw thread to the corresponding point on the next thread measured parallel to the axis.

Angle of Thread--The angle included between the sides of the thread measured in an axial plane. The thread angle of the American National Screw Thread form is 60 degrees.

Helix Angle--The angle made by the helix of the thread at the pitch diameter with a plane perpendicular to the axis.

RELATED
INFORMATION

BEST COPY AVAILABLE

TITLE: Screw Thread Formulas

UNIT: Screw Threads

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the formulas used in making calculations for cutting screw threads.

REFERENCE: Axelrod, Aaron. Machine Shop Mathematics. New York: McGraw-Hill Book Co., Inc.

INTRODUCTION: The following formulas apply to the three series, NC, NF and NS of the American National Screw Thread form.

SYMBOLS:

P --Pitch
SD--Single Depth
DD--Double Depth
C --Crest
R --Root
N --Number of threads per inch
TDS--Tap drill size

FORMULAS:

$$P = 1/N$$

$$SD = \frac{.6495}{N}$$

$$DD = \frac{1.299}{N}$$

$$DD = 2SD$$

$$C = p/8$$

$$R = P/8$$

Minor diameter = Major diameter - DD

Pitch Diameter = Major diameter - SD

TDS = Major diameter - 1/N

RELATED
INFORMATION

TITLE: The Rotary Table and Its Use

UNIT: Milling Machine

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the rotary table and its many uses.

REFERENCE: Burghardt-Axelrod-Anderson. Machine Tool Operation II. New York: McGraw-Hill Book Co.

INTRODUCTION: Rotary tables are integral equipment on slotters or vertical shapers and auxilary equipment for vertical milling machines and jig borers. Rotary tables are also used occasionally on horizontal boring mills, milling machines and drill presses.

INFORMATION: There are only four fundamental applications. One is the machining of circular contours and grooves. Another is the locating and finishing of surfaces at an angle to some other angle. The third application is boring holes at angular locations. Finally, the rotary table is employed for divisional spacing like the index head.

Most rotary tables can be fastened at any position on the machine table. A few have auxiliary powerfeed connections that permit only limited freedom. Most tables rely on the handwheel for the source of rotary movement. On the vertical shaper and slotter the rotary section is integral. It has been substituted for the table and fitted to the saddle by the cross-slide. In this way, feed in three directions can be obtained; front to rear, crosswise and circular. Vertical shapers likewise usually have powerfeed in these directions.

Sometimes "circular" tables, as they are sometimes called, are clamped to tall angle plates or knees. Tables are most often used in this way or the horizontal boring mill. When settings other than 90 degrees to the base are required, some plates are placed under them.

Rotating tables have relatively simple mechanisms. Commercial sizes are 9", 12", 15", and 18". Larger tables of different sizes are usually supplied directly by the machine manufacturer.

In many cases these are adaptable only to one machine. Provisions is made on most tables to "throw out the worm" in order to make quick adjustments manually. Some have the handwheel replaced by a removable crank. A few are made so that the familiar index head dials may be attached.

BEST COPY AVAILABLE

RELATED INFORMATION

TITLE: Operation of Shaper

UNIT: Shaper

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the type and size, methods of mounting work on the shaper, adjustment of the length of stroke, and speed and feeds to be used.

REFERENCES:
1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. McCarthy-Smith. Machine Tool Technology. Bloomington, Illinois: McKnight and McKnight Publishing Co.

INTRODUCTION: Shapers are primarily intended for machining flat surfaces but can be used for machining many kinds of curved, odd, or irregular surfaces.

INFORMATION:

1. Types and sizes of shapers
   - A. Types
     - (1) Horizontal
     - (2) Vertical
   - B. Sizes are determined by maximum length of stroke

2. Methods of mounting work
   - A. Work is held in vise when possible. Parallel bars are placed under the work to properly position for machining.
   - B. Work may be clamped directly to the table.

3. Adjustment of stroke
   - A. The length of stroke is the distance the tool moves in one direction for any given time. The proper length of stroke for any given job is the length of the work plus 2 inches. Various lengths of strokes are available on shapers.

4. Selecting and setting proper speeds and feeds
   - A. The speed to use on any given job can be determined through calculation using the following formula:

     $$S.P.M. = \frac{6(CS)}{L}$$

     C.S. = Cutting Speed

     L. = Length of stroke

     (90'/M for steel)
   - B. Feed is expressed in thousandths of an inch per stroke
     - (1) Roughing cuts use .040"-.050" per stroke.

(2) Finishing cuts use .010"-.020" per stroke.

5. The cutting tool should be held at right angles to the face being machined. When "down feeding" the cutting tool should be placed so that it projects beyond the sides of the clapper box and approximately 45 degrees with the face being machined.

RELATED
INFORMATION

BEST COPY AVAILABLE

'TLE: Shaper Operations

IT: Shaper and Shaper Processes

'CUPATION: Machinist

JECTIVE: To acquaint the student with different operations on a shaper that may be accomplished on the shaper.

FERENCE:

TRODUCTION: The terms parting, cutting off, slitting, slotting and grooving are often used synonymously and sometimes rather loosely. Their definitions are not established and the machinist may have some trouble making a distinction between them.

FORMATION: Parting-Parting or cutting off is the process of separating or cutting off material. Although it is possible to cut within a few thousandths of the desired length, parting is not usually considered an accurate process. The narrow parting or cutting off tool is fed down vertically into the material, and each succeeding stroke of the tool cuts deeper into the metal until the parts are separated.

Slitting-Slitting usually implies a narrow cut. The cut may be of any length, deep or shallow. Slits should not be too deep because of the difficulty encountered when a deep slit is being cut with a narrow tool. Slitting therefore, is machining a narrow cut not over 3/16' wide and does not complety sever or cut off the metal.

Slotting-Slotting is usually understood to indicate an opening that is more than 3/16" wide. A slot may have one end open, both ends open or both ends closed. The sides of the cut may be straight or sloping. The slot may be cut in one operation with a tool cut to the desired wideth of the slot. If the slot is wider than the tool, the tool can be set to cut down one side of the slot, then raised and set to feed down to complete the second side of the slot. If the slot is more than twice as wide as the tool, the center can be cut before cutting down the sides. If the slot is unusually wide and deep, a number of cuts can be taken with a wide tool to remove the excess metal, and then the sides an [illegible] with side cutting tools.

Keyways-Keyways are slots which are cut to a standard width and depth to receive rectangular blocks or keys.

Grooving-Grooving should be considered the process of cutting a shallow slot. Such cuts may be square, rectangular, v-shaped or circular. Grooves can act as reservoirs for oil or for channels for the distribution of lubricant, or they may act as channels which aid in disposing of dirt or clips. Sometimes a groove is cut next to a shoulder as an aid in grinding.

Serrating-Serrating is the process of cutting a series of equally spaced grooves upon the surface of a work piece. The main reason for serrating is to roughen the surface slightly and to increase its holding power, although it is frequently used for ornamental purposes. One special use in the machine shop is to lessen the effective area of a working surface, such as a lapping plate. The grooves may be cut perpendicular to each other. The plane surface bounded by the grooves will then be rectangular or square. If the grooves cross each other at an angle other than 90 degrees, the area will be diamond shaped.

While the shapes of the grooves are not standarized they are usually v-shaped with a flat or rounded surface at the bottom of the two tapered sides. The depth and distance between the grooves determine the size of the flat surfaces. Lard oil or other suitable coolant, applied with a brush, will help the cutting action and improve the finish.

RELATED
INFORMATION

TITLE: Spur Gear Terminology

UNIT: Milling Machine

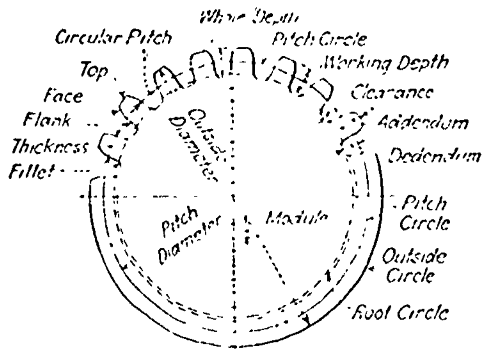
OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the definitions and terms that are common to spur gears.

REFERENCE: Anderson-Tatro. <u>Shop Theory</u>. New York: McGraw-Hill Book Co., Inc.

INTRODUCTION: The machinist must be familiar with the terms used to describe gears, so he will understand the methods used for designing, calculating and cutting gears.

INFORMATION: Parts of a Spur Gear:



Spur-gear elements and tooth parts.

TERMS:

1. Spur Gear--A gear with teeth cut parallel to the axis of rotation.
2. Addendum--The distance from the pitch circle to the top of the tooth.
3. Dedendum--The distance from the pitch circle to the bottom of the tooth.
4. Outside Diameter--The over-all diameter of the gear.
5. Pitch Diameter--The diameter of the pitch circle.
6. Pitch Circle--The circle made by the line of contact of two cylinders which would have the same speed ratios as the gears.
7. Circular Pitch--The distance from one point on a gear tooth to the corresponding point on the next tooth, measured on an arc of the pitch circle.

8. Clearance--The distance between the end of a tooth and the bottom of a mating tooth.
9. Working depth--Distance a tooth extends into the tooth space when fully meshed and has correct clearance.
10. Whole depth--Working depth plus the clearance.
11. Diametral Pitch--Number of teeth per inch of pitch diameter.
12. Tooth face--Surface of the tooth from the pitch circle to the outside.
13. Tooth flank--Surface of the tooth from the pitch circle to the base of the tooth.
14. Chordial thickness--Thickness of a tooth where the pitch circle passes through the tooth.
15. Center distance--The distance is the measurement from the center of one gear to the center of a meshing gear.
16. Root circle--A circle formed by the bottom of the teeth.
17. Rack--A gear rack is a piece of material with teeth cut on a flat surface.
18. Pressure angle--The angle at which pressure of one tooth upon another is applied.

RELATED
INFORMATION

TITLE: Spur Gear Formulas

UNIT: Milling Machine

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the formulas used to make spur gear calculations.

REFERENCE: 1. Anderson-Tatro. Shop Theory. New York: McGraw-Hill Book Co., Inc.
2. Burghardt-Axelrod-Anderson. Machine Tool Operation Part II. McGraw-Hill Book Co., Inc.

INTRODUCTION: To be able to make the calculations needed to design and cut a spur gear, a machinist must be familiar with the symbols and formulas.

INFORMATION: SYMBOLS:

P= Diametral Pitch
CP= Circular Pitch
PD= Pitch Diameter
OD= Outside Diameter
N= Number of teeth
S= Addendum
S+F Dedendum
F= Clearance
W= Whole depth
T= Chordial thickness
LP= Linear Pitch
H= Corrected addendum
NG= Number of teeth on gear
NP= Number of teeth on pinion
C= Center distance
I= Inside diameter
L= Length of rack

FORMULAS:

$S = 1/P$	Addendum
$C = \frac{Ng + Np}{2P}$	Center distance
$CP = \frac{3.1416}{P}$	Circular Pitch
$F = \frac{.157}{P}$	Clearance

$S + F = \frac{1.157}{P}$	Dedendum
$P = \frac{3.1416}{CP}$	Diametral Pitch
$P = \frac{N + 2}{O\ D}$	Diametral Pitch
$P = \frac{N}{PD}$	Diametral Pitch
$N = P \times PD$	Number of teeth
$OD = \frac{N + 2}{P}$	Outside diameter
$OD = PD + 25$	Outside diameter
$PD = \frac{N}{P}$	Pitch diameter
$PD = OD - 25$	Pitch diameter
$W = \frac{2.157}{P}$	Whole depth
$W = .6866 \times CP$	Whole depth
$LP = \frac{3.1416}{P}$	Linear Pitch

RELATED
INFORMATION

ITLE: Iron and Steel Products

NIT: Heat Treatment and Testing of Materials

CCUPATION: Machinist

BJECTIVE: To acquaint the student with the different processes of producing steel.

EFERENCE: <u>Tool Steel Simplified</u>. Carpenter Steel Co.

NTRODUCTION: The making of steel and the process in which it is produced is necessary to the machinist in order that he may have a better understanding of what he is working with and how to select the right steel for the job.

NFORMATION: Cast iron goes through a heating and refining process in a "cupola furnace' which is somewhat less complicated than the blast furnace. Air at normal temperature is forced through a burning mixture of pig iron, scrap, limestone and coke. Cupola temperatures are lower than those of the blast furnace, and the refining or purifying process can be regulated easily for its carbon content. Any addition to cast iron are usually added to the stream as modifying elements. The most common are nickel and chromium.

STEEL CASTING PRODUCTION
In the steel foundry, corrections are usually added to the hearth just before the heat is considered ready. The time duration of the two boiling processes, namely that of the ore and the lime, reduce nearly all minor alloying elements to a satisfactory level. Carbon, the major element besides iron, continues to be reduced during a third period of the heat until a satisfactory fracture test has been observed by the melter.

STEEL PRODUCTION PROCESSES
The bulk of steel production is poured not for castings but into long tapered square shaped molds called ingots. Ingots are an intermediate process, similar to that of casting pig iron, between chemical metallurgy and mechanical metallurgy. Castings, when they are poured, retain their absorbed and trapped gases to some degree even when properly vented. Ingot molds are designed to lead impurities and gases toward the top of the cast where they can be removed easily by a process called "cropping."

When ingots have been prepared properly the steel is reduced to a designated shape through rolling, forging, pressing or extrusion. The rolling of flat material to thinner thickness is referred to a rolling strip. When strip steel is slitted into narrow widths and rolled into coils it is known by either strip or coil Wide strips are usually cut to specific lengths for convenience in handling.

When these strips are wide and also thick, they are often called plates. Similar sections of tool steel that are thicker than the number sizes listed for gaging sheet steel are often called flats. When cut into convenient lengths, the lengths are called bars.

COLD WORKING OF STEEL

Hot working of steel can produce finished shapes within specified limits of accuracy. Cold working processes continue to compress and strengthen material from which the scale has been removed. With no heat applied, cold rolling or drawing becomes a finishing process both as to surface and as to dimensional limits. Machinists should know the capabilities, limitations and strains set up within cold worked materials. Strains offer little trouble if cuts are balanced so that equal cuts remove equal stress from opposite sides.

TOOL STEELS

Tool steels with the exception of drill rod have a decarborized surface, even though care is taken during the annealing process. Tool steels are always annealed because of the work hardening effect mentioned earlier. A surface cut to specified depths according to size of material should always be taken to remove surface material not likely to harden. In similar maner, tool steels which have been forged require annealing before accurate cuts may be made.

RELATED
INFORMATION

TLE: Manufacture of Steel

IT: Heat Treatment and Testing of Materials

[illegible]ON: Machinist

JECTIVE: To acquaint the student with the different processes of producing steel.

FERENCE: 1. Tool Steel Simplified. Carpenter Steel Co.

TRODUCTION: The word "steel" is a most inclusive term which in shop language, often refers to metal in the iron-steel family.

FORMATION: Steel is pig iron refined to specifications regarding purity and strength. Steels are, therefore, often, classified according to charical composition. Plain steels have carbon as the alloying element. Ranges of carbon are: Low (0.10-0.30), Medium carbon steels (0.40-0.70) and the hardening variety (0.80-1.50), often called tool steel.

Steels are frequently named according to the processing they have passed through. Some of these names are: electric furnace, open hearth, Beasemer and crucible. Other steel names, depending on use are structural, spring, rail, boiler plate, armor plate, free cutting, screw machine stock and others.

Common shapes are plates, sheets, strip, wire and bar. Mechanical processing often utilizes specific kinds of steel. Hot rolled, cold rolled, cold drawn, forging, extrusions, machine, electric, drill rod and high speed are terms which are associated with certain qualities and conditions.

Each metal has its own set of specifications, recognized appearance and qualities, working conditions and limitations. Finding out how and where each steel is made, used and worked is important because steel is an essential metal in production.

OPERATION SHEET

BEST COPY AVAILABLE

TITLE: To Grind A General Purpose Turning Tool

UNIT: Lathe

OCCUPATION: Machinist

OBJECTIVE: To give the student practice in grinding a general purpose turning tool bit.

INFORMATION: The proper grinding of lathe tools is an operation with which the lathe operator must become familiar. The angles at which a tool bit performs best should be noted, and these should be duplicated on cutting tools of the same type. The importance of relief (clearance) and rake angles should not be overlooked.

REFERENCE: ...r and Oswald. Turning Technology. New York: Delmark Publishing Co. Chapter 6.

PROCEDURE:

1. Grasp the toolbit firmly and support the hands on the grinder tool rest.

2. Adjust the position of the toolbit blank and grind the cutting edge. Step no. 1.

3. At the same time, tilt the bottom of the toolbit in and grind the 6° - 8° side-relief angle.

4. Continue grinding until the side cutting edge is approx. 7/16" to 1/2" long and the point is about 1/4 the width of the toolbit. Step no. 2.

5. Cool the toolbit frequently in water so that it does not overheat during the grinding operation. Over-heating may damage the toolbit. NOTE: Stellite and cemented-carbide toolbits should never be quinched when being ground.

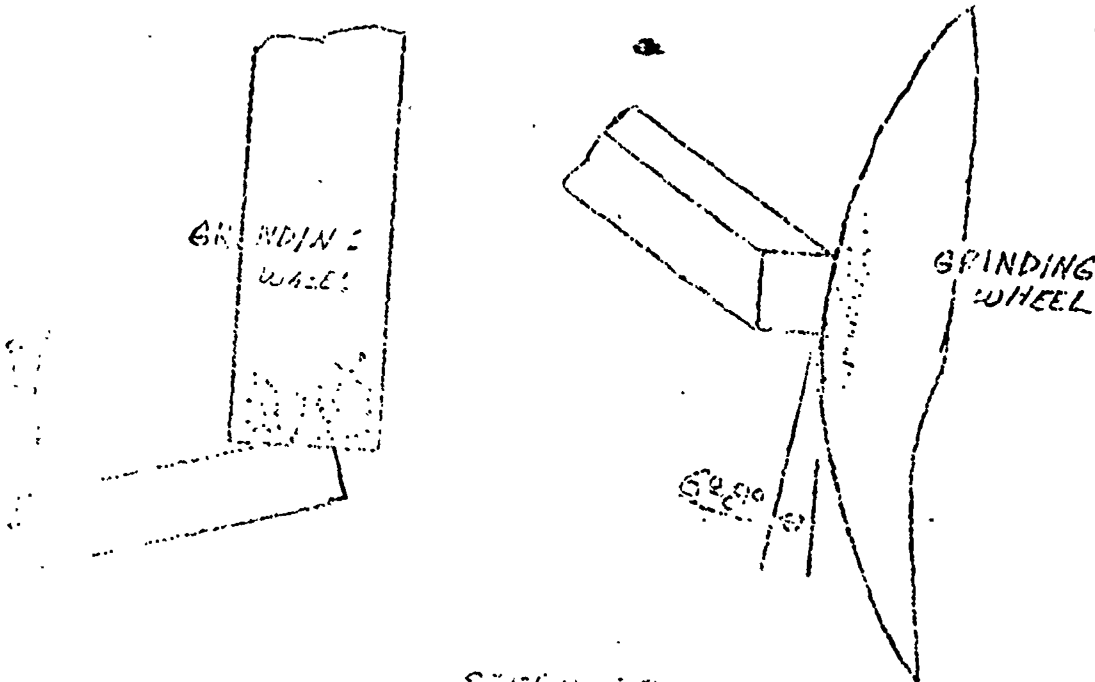
6. Grind the end cutting edge so that it forms an angle of less than 30° with the side cutting edge. Step no. 2. The toolbit should be held so that the back end is lower than the point. This forms the end-relief angle of 22° at the same time.

7. Round the points slightly, maintaining the same end and side relief angles.

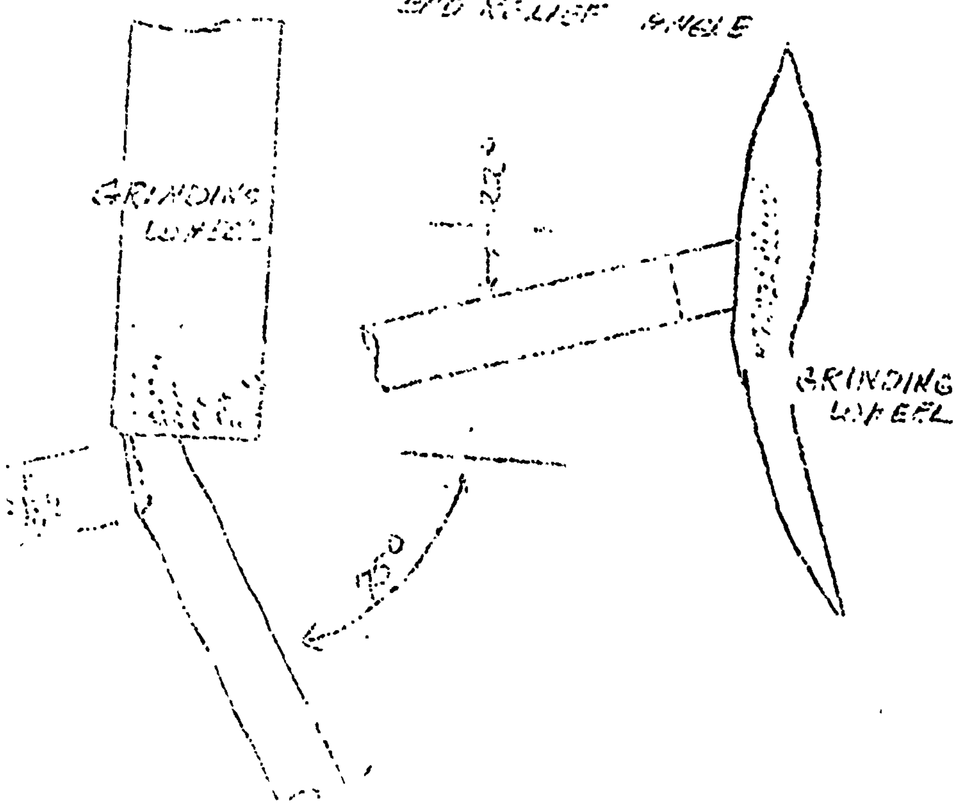
8. Hold the toolbit so that it is approximately 45° to the axis of the wheel and tilt the bottom of the tool bit in so that the side rake of about 10°-12° is ground on the top of the toolbit. Step no. 4.

9. Hone the cutting edge and the point of the toolbit with a fine oilstone. This produces a keener edge and a better surface, and increases the life of the toolbit.

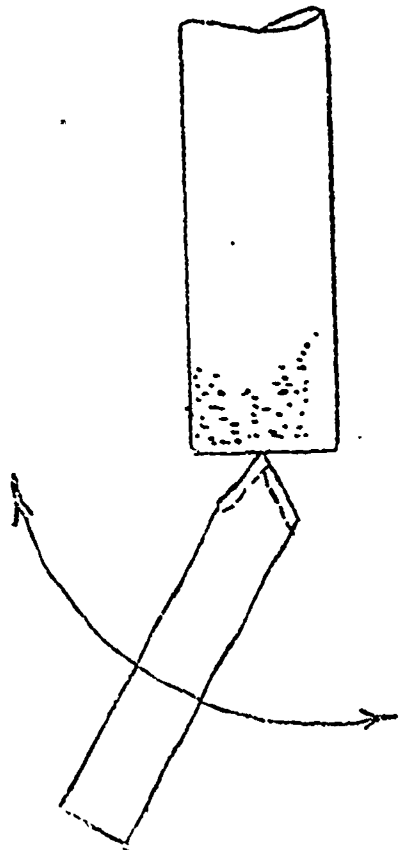
STEP 1 SIDE ANGLE



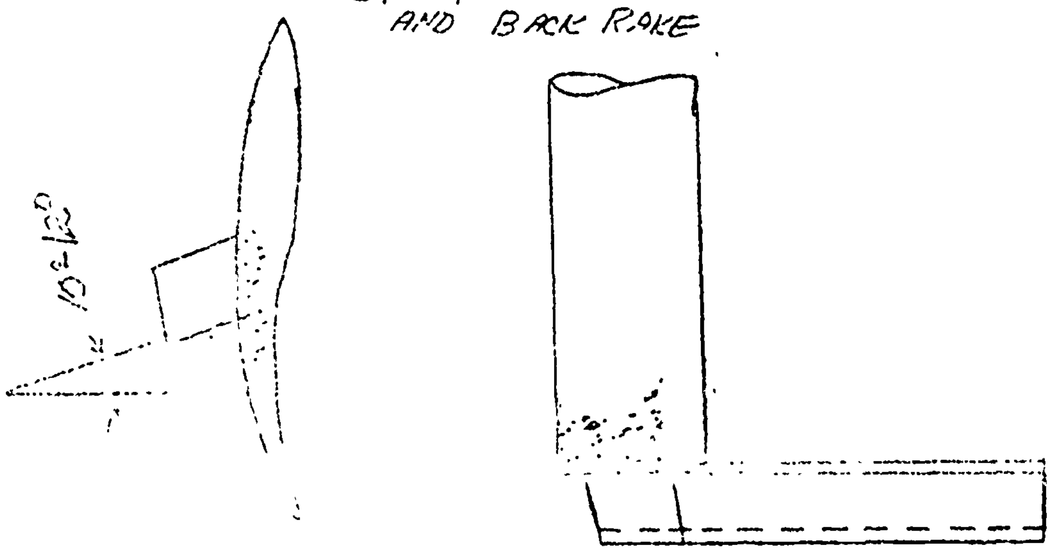
STEP 2 GRIND END RELIEF ANGLE



STEP 3 GRIND 1/32 RADIUS
OR TO SUIT JOB



STEP 4 GRIND SIDE
AND BACK RAKE



OPERATION SHEET

BEST COPY AVAILABLE

TITLE: To Knurl Work in the Lathe

UNIT: Lathe

OCCUPATION: Machinist

OBJECTIVE: To give the student practice in the proper procedure for knurling.

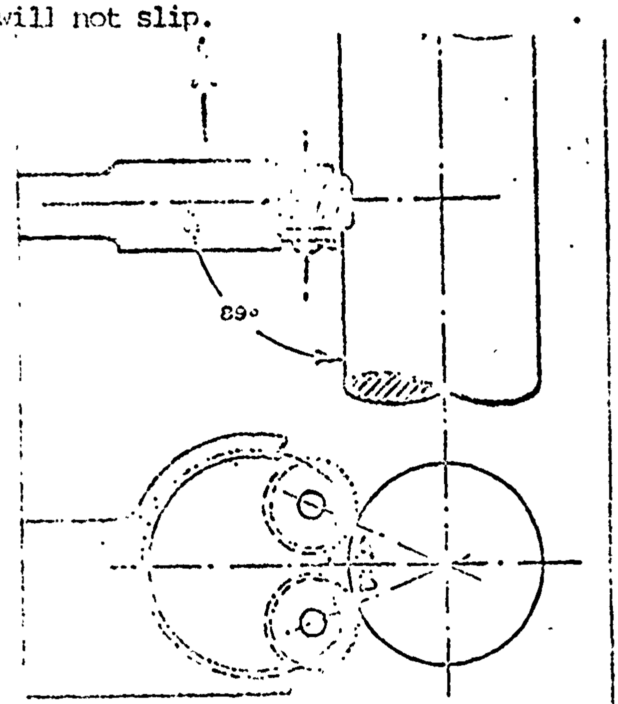
INTRODUCTION: Many tools and machine parts have knurled surfaces for ease of gripping. Knurling is done by embossing a regular diamond shaped pattern on the surface of work, revolving in the lathe, with a pair of rollers having serrated surfaces. There are three grades of knurling rollers, course, medium, and fine.

REFERENCE: Nicholson, Fred. Shop Theory: New York: McGraw-Hill Book Co. Inc. - Chapter 14.

PROCEDURE:

1. Locate the limits of the Knurl on the surface of the work.

2. Set the Knurling tool so that the top roller is the same distance above the center of the work as the bottom roller is below the center of the work. The serrated faces of the Knurling rolls are set parallel to the surface of the work. (See Fig. 1). Tighten the tool post screw so that the knurling tool will not slip.



ANGLE A = ANGLE B

3. Set the speed of the lathe below 100 RPM.

4. Feed the lathe at .015" per revolution.

5. With the hand feed control, move the knurling tool into position at the right end of the portion to be knurled.

6. Start the lathe.

7. Force the knurling rolls into the work with the hand cross feed to a depth of .010 - .015 of an inch.

8. Engage the feed clutch and let the knurling tool traverse the surface to be knurled.

   CAUTION: Use plenty of cutting oil on the surface that is being knurled.

9. When the tool reaches the end of theportion to be knurled, stop the lathe. Do not back the tool away from the work or disengage the feed clutch.

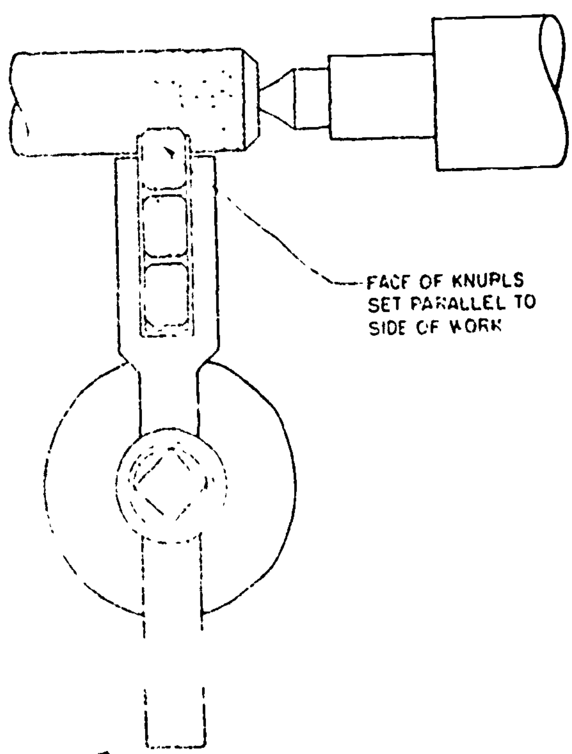
   CAUTION: If the tool is backed out or the clutch disengage the knurling rolls will not track on the next pass.

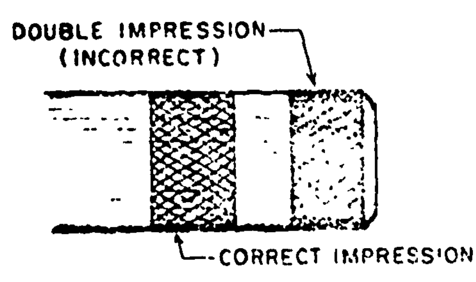
10. If the knurl is not deep enough reverse the direction of the feed.

    CAUTION: Always stop the lathe before reversing the lathe feed.

11. Start the lathe. Force the knurling rolls into thework another .010 - .015 of an inch. Apply plenty of cutting oil to the surface of the knurl.

12. The above steps may be repeated if needed to produce the desired depth of knurl.





BEST COPY AVAILABLE

[illegible] To Grind a Drill

[illegible] Grinding

[illegible] Machinist

[illegible] To develop the skill needed to sharpen a twist drill by hand.

[illegible] To cut properly a drill must be sharpened to the correct point angle (See Fig. 1) and the cutting lips must have the proper clearance angle (See Fig. 2). The angle of the two cutting lips must be the same and length of the cutting lips must be equal. With care small drill may be sharpened by hand on the bench grinder with a 60 grit aluminum oxide grinding wheel.

[illegible] Ludwig, O. A. Metal Work Technology and Practice. Bloomington, Ill. McKnight and McKnight Pub. Co. Unit 26

[illegible] 1. Start the grinder.
Safety Note: Be sure to wear goggles when grinding.

2. Hold the drill so that the cutting lip is parallel to the face of the wheel. Grind lip until bright metal shows all the way across.
(See Fig. 2)
Caution: When grinding be careful not to get the drill hot enough that it changes color as this will destroy the temperature of the cutting edge. Dip the drill point in water frequently to help keep it cool.

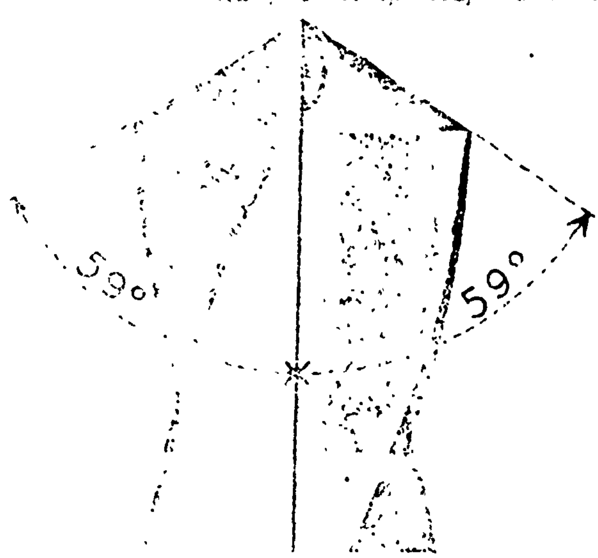


Fig. 1

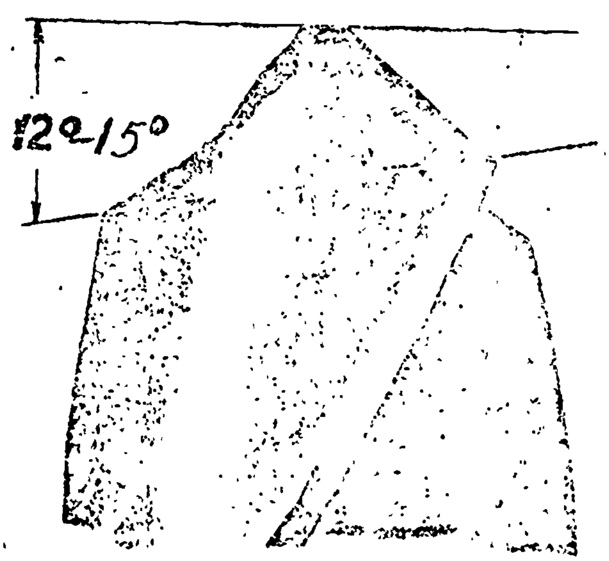


Fig. 2

3. Check each lip with the drill point gage. If the angles are [illegible] lips are not of equal length, regrind until [illegible] length of the lips are correct.
(See Fig. 3).

4. Grind the lip clearance, 12 to 15 degrees on one lip. Hold

the drill with the rake angle up against the wheel and rotate the drill to the right, pressing it against the wheel so that the grinder removes metal from behind the cutting lip. (See Fig.5)

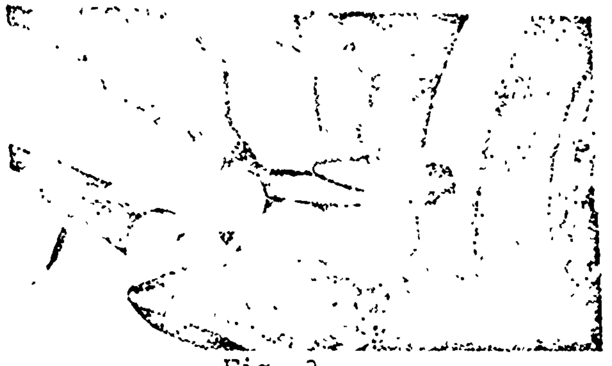


Fig. 3

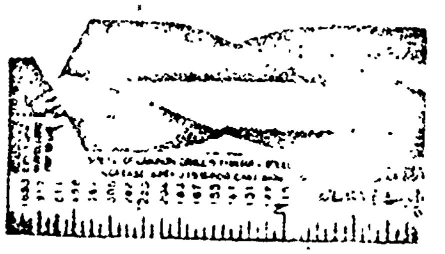
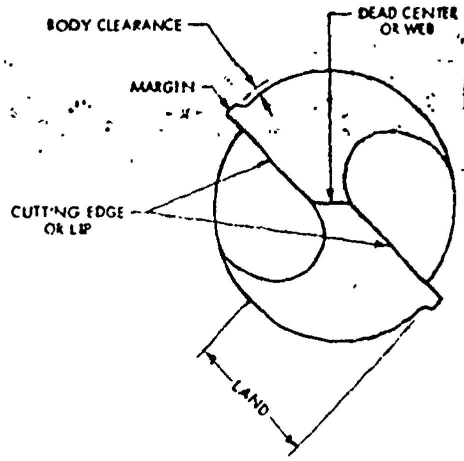


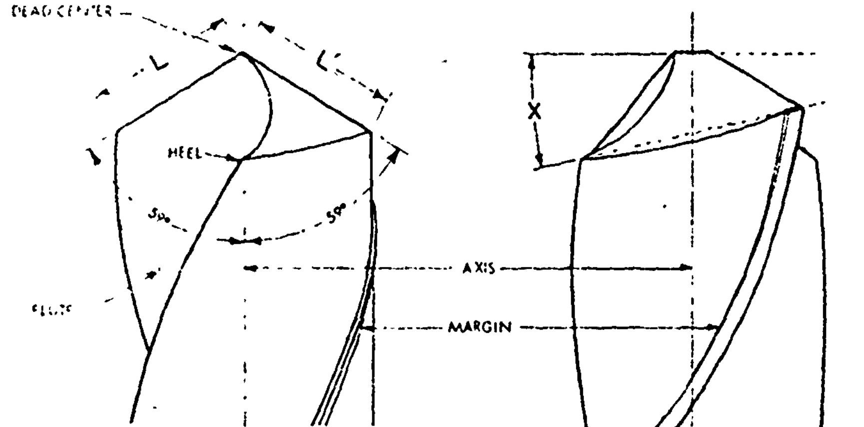
Fig. 4

6. Use the same procedure for grinding the lip clearance on the opposite lip.

7. Recheck to see that the lips are the same angle and that the lips are the same length. If not regrind until they are.

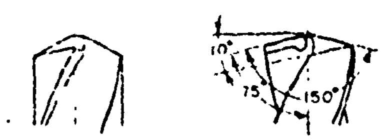
Fig. 5



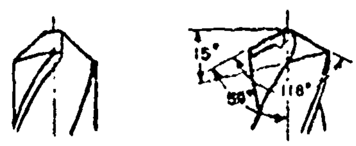


(A)
BOTH LIPS MUST BE AT THE SAME LIP ANGLE AND OF EQUAL LENGTH

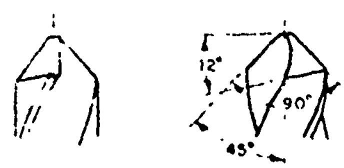
(B)
LIP CLEARANCE X



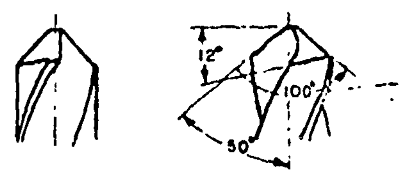
HARD MATERIALS AND STEEL RAILS



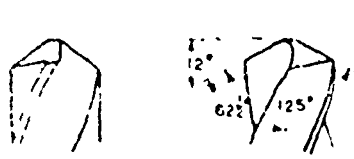
SOFT BRONZE AND BRASS



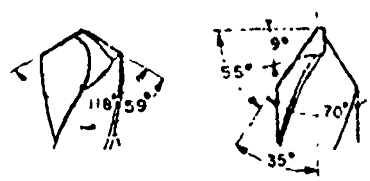
MAGNESIUM ALLOYS, CAST IRON, AND DIE CASTINGS



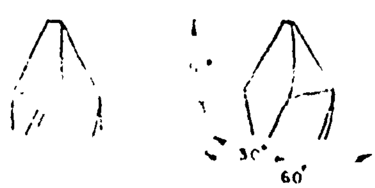
COPPER ALLOYS AND COPPER



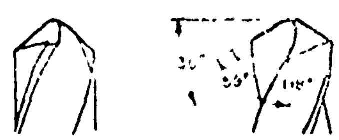
... AND HEAT-TREATED STEEL



DEEP-HOLE DRILLING AND CRANKSHAFTS



ALUMINUM, HARD RUBBER, FIBER, AND WOOD



MOLDED MATERIALS AND PLASTICS

OPERATION SHEET

BEST COPY AVAILABLE

TITLE: Thread Cutting

UNIT: Lathe Work

OCCUPATION: Machinist

OBJECTIVE: To acquaint the student with the proper procedure for setting up calculating and cutting machining a screw thread on the lathe.

REFERENCE: 1. Anderson-Tatro. Shop Theory. New York: McGraw Hill Book Co., Inc.

INTRODUCTION: To operate the lathe successfully you as a student machinist should set tools and make necessary machine adjustments to familiarize yourself with the operation.

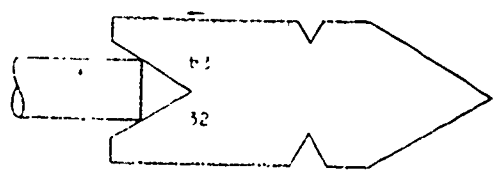
INFORMATION:

1. The single depth of the thread should be calculated by the formula suited to the kind of thread to be cut.
2. Pitch x 3/4 will give the amount to feed in on the compound.
3. Set the compound slide 30 degrees to the right (for right hand threads).
4. Set the tool bit square with the axis of the work using the center gage and on the center line of the work.
5. Gear the lathe for the required number of threads per inch to be cut.
6. Mesh the worm gear of the chasing dial with the lead screw and determine which of the lines are to be used.
7. Start thelathe and touch the cutting tool to the revolving work. Set the graduated dials on the cross feed and compound slide to zero.
8. Most lathes have an adjustable stop to prevent feeding the tool too far into the work on successive cuts. Set the stop at this point.
9. Move the cutting tool off the work a short distance from the end so it is in the clear. Feed the tool .002" to .004" deep using the compound slide. It is better to feed the tool in on successive cuts with the compound slide. The tool cuts on one side only and produces a smoother thread. The cross feed screw is used to pull the tool andre-set the tool against the adjustable stop after each cut.
10. With the lathe running, engage the half nut at the correct selected line on the chasing dial and make the first cut.
11. Withdraw cutting tool and disengage half-nut from the lead screw. Return the carriage to the starting position by hand feed.

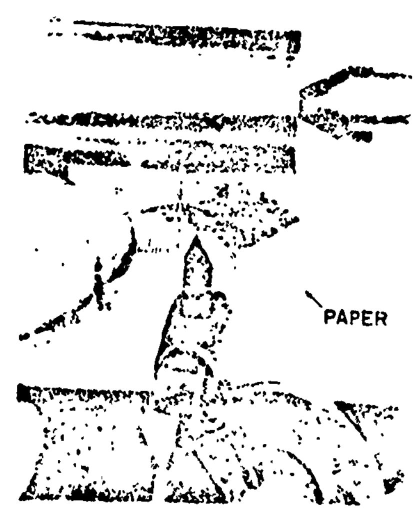
12. Check to see that the correct number of threads per inch are being cut.

13. Take successive cuts by feeding the tool in .002" to .003" per cut. Use cutting oil on the tool bit for a smoother thread.

14. When the thread is cut nearly to the correct depth use a thread ring gage or a nut check the fit, depending upon the degree of accuracy required. Precision threads may be measured by the three-wire method. A finished screw thread should have the end chamfered.

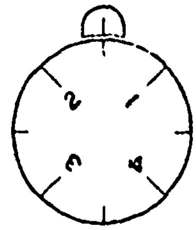
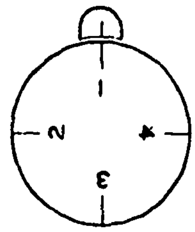
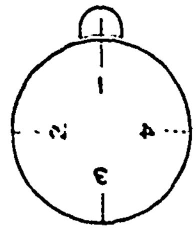


A CENTER GAGE IS USED TO CHECK TH ANGLE OF A 60° THREADING TOOL.

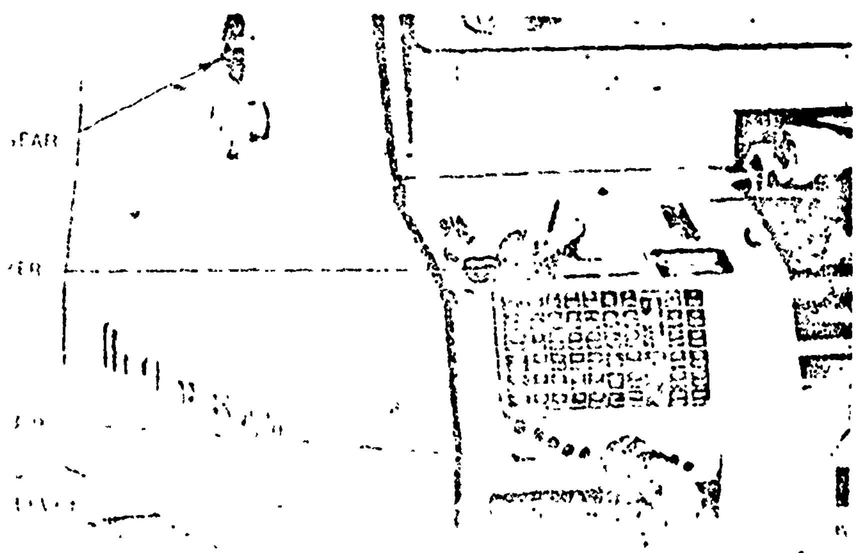


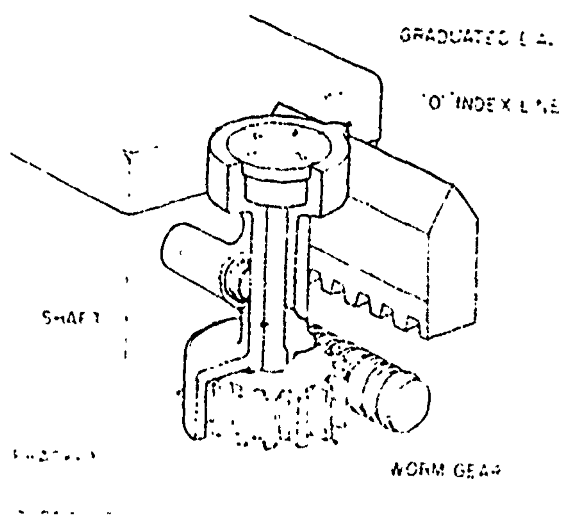
A THREADING TOOLBIT GROUND FO USE IN A LEFT HAND OFFSET TOOLHOLDER

BEST COPY AVAILABLE

THREADS PER INCH TO BE CUT	WHEN TO ENGAGE SPLIT NUT		READING ON DIAL
EVEN NUMBER OF THREADS	ENGAGE AT ANY GRADUATION ON THE DIAL	1 1 ½ 2 2 ½ 3 3 ½ 4 4 ½	
ODD NUMBER OF THREADS	ENGAGE AT ANY MAIN DIVISION	1 2 3 4	
FRACTIONAL NUMBER OF THREADS	1/2 THREADS, E.G. 11 1/2 ENGAGE AT EVERY OTHER MAIN DIVISION 1 & 3, OR 2 & 4 OTHER FRACTIONAL THREADS ENGAGE AT SAME DIVISION EVERY TIME		
THREADS WHICH ARE A MULTIPLE OF THE NUMBER OF THREADS PER INCH IN THE LEAD SCREW	ENGAGE AT ANY TIME THAT SPLIT NUT MESHES		USE OF DIAL UNNECESSARY

RULES FOR ENGAGING THE SPLIT-NUT FOR THREAD CUTTING





OPERATION SHEET

BEST COPY AVAILABLE

TITLE: Internal Threading on a Lathe

UNIT: Screw Thread processes

OCCUPATION: Machinist

OBJECTIVE: To give the student practice in the proper procedure for internal threading on the lathe.

INTRODUCTION: Many internal threads are finished with a tap; however, if a certain size of tap is not available, or when it is essential that the thread is concentric with the diameter, it must be cut on the lathe.

REFERENCE: Fraser and Oswald. Turning Technology. New York: Delmar Publishing Co. Chapter 29.

PROCEDURE:

1. Calculate the tap drill size.

2. Mount the work in a collect, chuck, or on a faceplate.

3. Drill a hole approximately 1/16" smaller than the tap drill size.

4. Set up a boring bar and bore the hole to calculated tap drill size.

5. Counterbore the end of the hole to themajor diameter of the thread for a distance of about 1/16". This serves as a guide to the depth of thread during the threading operation.

6. If the hole to be threaded does not go through the metal (a blind hole), it is necessary to cut a recess at the end of the thread to provide clearance for the threading tool at the end of the cut. This recess should be cut slightly deeper than the major diameter andshould be wide enough to permit the toolbit to clear the thread when the splitnut is disengaged.

7. Set the compound rest to 29° to the left.

8. Set the quick-change gearbox for the correct number of threads per inch.

9. Engage the lead screw.

10. Mount a threading tool in the boring bar and set the point of the tool on center. The boring bar should be parallel to the centerline of the machine.

11. Square the threading tool with a center gage.

[illegible] the hole to be threaded should be marked on the boring bar, measuring from the threading tool. When this mark is even with the left edge of the workpiece, the split-nut lever must be disengaged.

BEST COPY AVAILABLE

13. Start the lathe and turn the crossfeed handle until the point of the toolbit touches the diameter of the work.

14. Set the crossfeed and compound rest graduated collars to zero and clear the cutting tool from the hole with the carriage handwheel.

15. Turn the compound rest handle counter-clockwise to move the toolbit .003 - .005, and take a trial cut.

16. At the end of each cut (when the mark on the boring bar is even with the edge of the work piece), disengage the split-nut lever and turn the crossfeed handle clockwise to clear the toolbit from the thread.

17. Move the carriage to the right until the toolbit is clear of the work.

18. Check the pitch of the thread with a screw pitch gage.

19. Return the crossfeed handle to zero and set the depth of cut by feeding the compound rest counter-clockwise, about .010 to .015".

20. Take successive cuts, decreasing depths until the thread is to the proper depth. If the point of the threading tool is the correct width, the amount of compound rest feed can be calculated by applying the following formula:

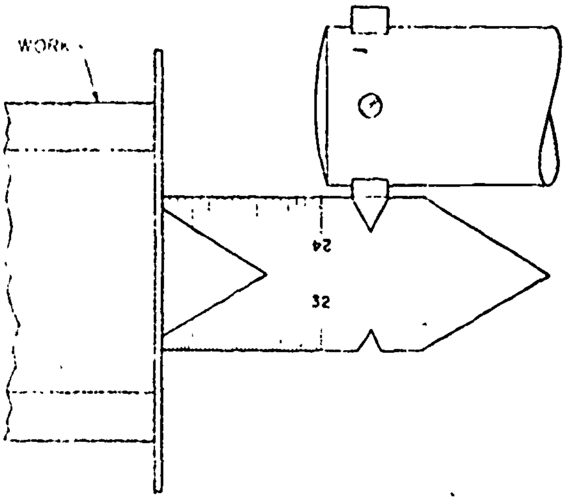
$$\text{Compound rest feed} = \frac{.750}{N}$$

$$= \frac{.750}{10}$$

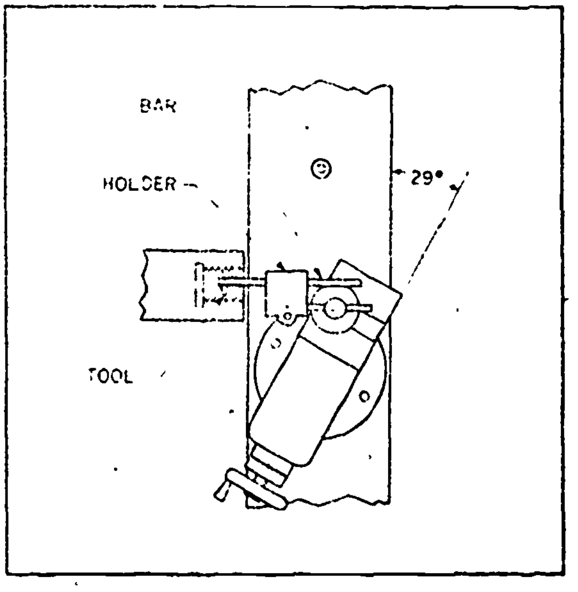
$$= \underline{.075}$$

21. As the thread becomes deeper, it is necessary to decrease the amount of compound rest feed to decrease the spring of the boring bar. The last few cuts should only be .001" deep in order to eliminate any spring remaining in the bar.

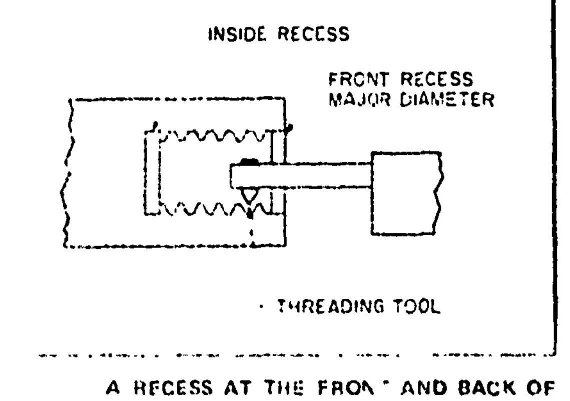
22. Check the thread for fit with a thread plug gage or a bolt.



BEST COPY AVAILABLE



... THE COMPOUND REST IS SWUNG TO 29° TO THE LEFT FOR CUTTING RIGHT-HAND INTERNAL THREADS.



A RECESS AT THE FRON AND BACK OF HOLE IS DESIRABLE WHEN INTERNAL THREADING.

BEST COPY AVAILABLE

OPERATION SHEET

TITLE: Spring Winding on a Lathe

UNIT: Lathe work

OCCUPATION: Machinist

OBJECTIVE: To give the student practice in winding coil springs on the lathe.

INTRODUCTION: One of the jobs often required of the machinist is the making of coil springs. If a spring winding machine is not available. In this operation, spring or music wire is wound around a mandrel of the proper diameter in an operation similar to left-hand thread cutting.

REFERENCE: Krar and Oswald. Turning Technology. New York: Delmar Publishing Co.

PROCEDURE:

1. Determine the correct wire size, mandrel size, and the number of coils per inch from Machinery's Handbook.

2. Mount the mandrel either between centers, in a drill chuck mounted in the head stock or in a 3-jaw chuck.

3. Set the lathe spindle speed to about 50 R.P.M.

4. Set the feed directional lever for left-hand threads.

5. Set the quick-change gearbox on the lathe to the required number of threads per inch. NOTE: After the spring has been wound and the tension is removed, the spring expands and elongates, so that it is larger with fewer coils per inch. It is therefore necessary to allow for this action when setting the machine. If six coils per inch are required on the finished spring, it is necessary to set the quick-change gear box to more than six, possibly seven, threads per inch.

6. Mount a wire guide block with the v-slot on the bottom in a left-hand offset lathe tool holder.

   NOTE: The guide block can be made from a piece of 5/16" or 3/8" square cold rolled steel with a shallow v-slot cut lengthwise to guide the wire. Tension can be applied to the wire by the tool holder screw.

7. Move the carriage until the end of the toolholder is opposite the hole in the mandrel.

8. Uncoil sufficient wire and feed it through the v-slot in the [illegible] into the hole of the mandrel.

9. Apply tension on the spring wire with the tool holder screw and then engage the split-nut lever.

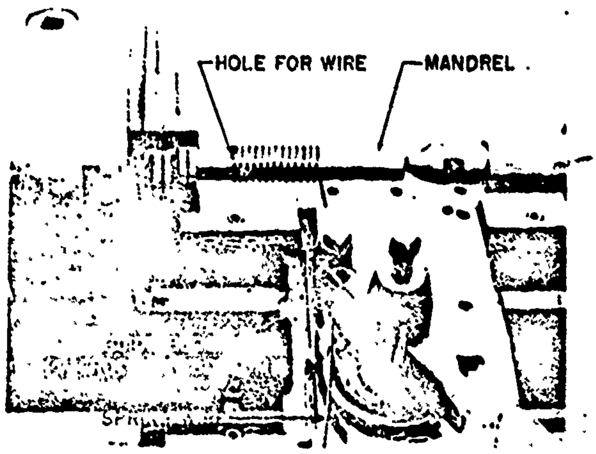
10. Start the machine and carefully let the wire feed through the block until the desired length of spring is wound.

11. Shut off the lathe and then disengage the split-nut lever. NOTE: If the spring must have close coils at either end, it is necessary to start the lathe and then engage the split-nut lever at the left end of the spring. When the spring is the desired length the split-nut lever is disengaged and the machine is shut off after sufficient close coils have been formed.

12. Release the tension on the spring by loosening the toolholder screw.

13. Using a pair of nippers, carefully cut the spring wire between the mandrel and the wire guide block.

14. Cut the left end of the spring with end nippers.



THE LATHE SET UP FOR ONE METHOD OF WINDING COIL SPRINGS

OPERATION SHEET

BEST COPY AVAILABLE

TITLE: To Harden and Temper Carbon Tool Steel

UNIT: Heat Treating

OCCUPATION: Machinist

OBJECTIVE: To give the student practice in hardening and tempering carbon tool steel.

INTRODUCTION: Hardening changes the physical properties of steel. Carbon is the element which causes steel to be hardened when it is heated to its critical temperature and quenched in oil or water. This will produce the maximum hardness in the steel. Tempering follows hardening to secure the degree of hardness of desired and to relieve the metal of strains and brittleness from hardening. In tempering, the steel is reheated to temperatures varying from 400 to [illegible] degrees Farhenheit, and cooled. When steel is tempered by this color method the following color chart may be used to determine the proper tempering temperature to use for the degree of hardness desired.

F°	Color	Use
380	Light yellow	Cutting tools for lathes
425	Light straw	Drills, reamers, milling cutters
465	Dark straw	Taps, dies, hacksaw blades
490	Yellowish brown	Hammer faces, rivet sets, wood chisels
525	Purple	Center punches, scratch awls
545	Violet	Cold chisels, knives and axes
590	Pale Blue	Wrenches, screw drivers, hammers

REFERENCE: Porter, H. W., Lawshe, C. H. and Lascoe, O. D. *Machine Shop Operations and Setups.* Chicago: American Technical Society. Ch. 12.

PROCEDURE:

1. Light and adjust the furnace.
2. Use tongs and place the work in the furnace.
3. Heat to the hardening temperature (1450 deg. F) which is denoted by a full red color.
4. Remove from furnace and quench in water or oil, according to the type of steel used.
5. Test for hardness with a file. If it is properly hardened the file will not cut it.
6. [illegible] from the surface of the hardened piece so the [illegible] color can be observed.
7. [illegible] the work in the furnace, holding the work so that it is visible. Watch the proper color to appear.
8. When [illegible] temperature has been reached, quench the piece in [illegible] or oil.

JOB SHEET

TITLE: TO MAKE A DRILL DRIFT

UNIT: BENCH WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of some of the more common types of bench, layout and power tools.

INFORMATION: A drill drift is used to remove taper shanked drills from drill sleeves and drill press spindels.

SPECIFICATIONS:

DRILL DRIFT NO. 2

MATERIAL: Cold rolled steel ½"x1" flat stockx6" long.

TOOLS AND EQUIPMENT: Powerhacksaw, drillpress, bench vise, combination square, surface plate, prick punch, scriber, ball peen hammer, dividers, center punch, layout ink, 10" flat double cut bastard file, 8" flat single cut smooth file, radius gage, ¼" drill bit, a 60 degree countersink and hand hack saw.

PROCEDURE:

1. Select stock as specified
2. Cut to length with power hack saw.
3. Hold work on the bench vise and remove burrs with file.
4. File one end square to layout from
5. [illegible]
6. Locate center of ¼" radius and prick punch where lines intersect.

7. Set dividers to $\frac{1}{4}$" radius and scribe the arc.

8. Scribe a $4\frac{1}{4}$" line from the end as per drawing

9. Scribe a line to form angular side as per drawing.

10. Locate center ofhole to be drilled on opposite end.

11. Prick punch at the pointwhere lines intersect.

12. Set dividers to $\frac{1}{4}$" radius and scribe the arc.

13. Center punch location of hole to be drilled..

14. Mount work on drill press and drill $\frac{1}{4}$" hole.

15. Countersink each side of hole 1/16" deep.

16. Grip work in vise for sawing.

    Note: Use soft vise jaws to prevent marring work.

17. Saw off corners of radius and the angular side with hand hacksaw.

    Note: Leave at least 1/32" of material along the line to preserve layout.

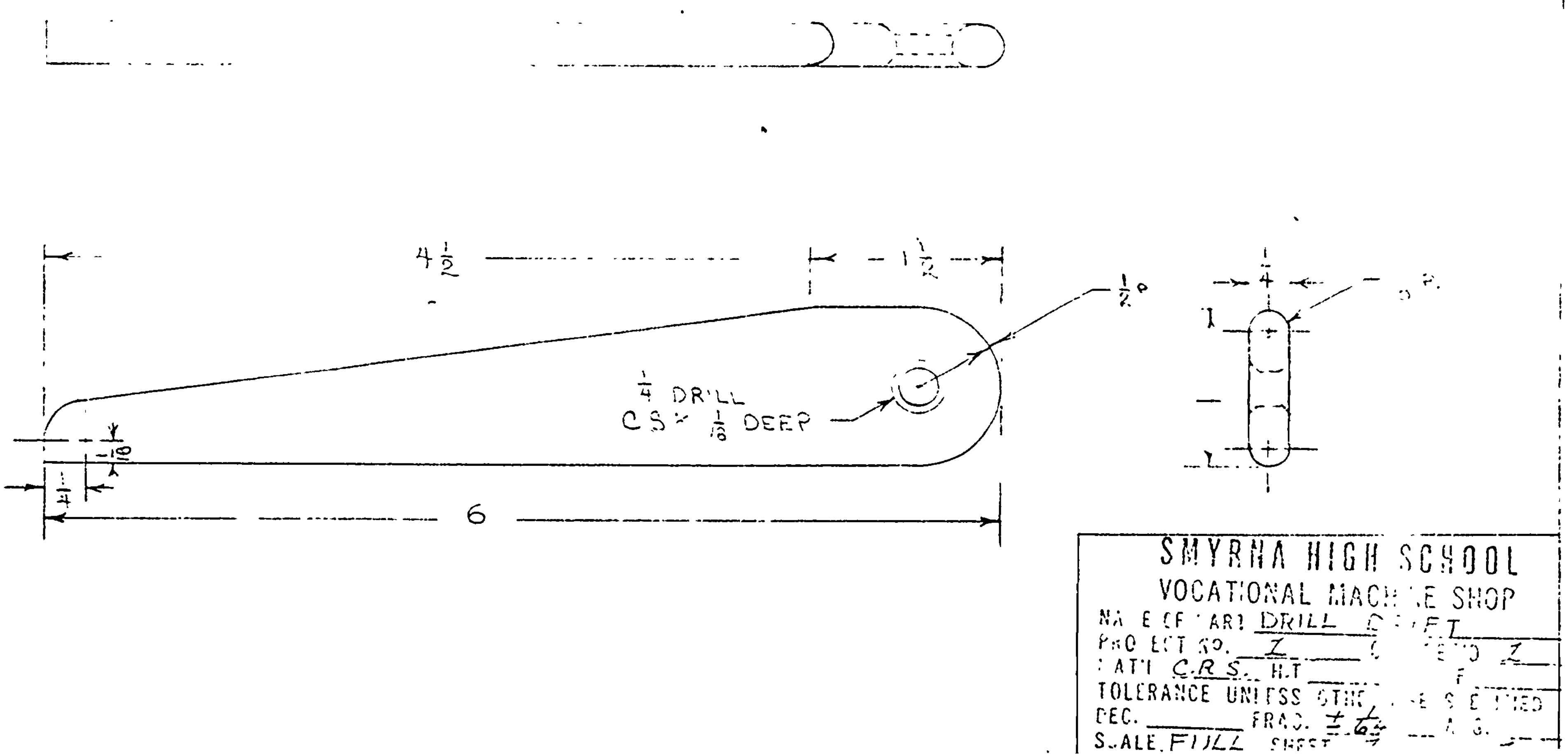
18. Rough file to line with a 10" bastard file

19. Rough file radius on edges as per drawing.

    Note: Check radiuson end and edges with a radius gage.

20. Finish file with a 8" single cut smooth file.

21. Inspect as per drawing.


4 1/2
1 1/2
1/2 R
1/4 DRILL
CS × 1/16 DEEP
6
1/4
SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
NAME OF PART DRILL DRIFT
PROJECT NO. Z
MAT'L C.R.S. H.T
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. FRAC. ± 1/64
SCALE FULL SHEET

JOB SHEET

TITLE: TO MAKE A TINNERS RIVETING HAMMER HEAD

UNTI: BENCH WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of the more common types of bench, layout and power tools.

INFORMATION: The riveting hammer is used in sheet metal work for heading rivets.

SPECIFICATIONS:

TINNERS RIVETING HAMMER

MATERIAL: Cold rolled steel 3/4" x 3/4" x 4 1./8"

TOOLS AND EQUIPMENT: Power hack saw, drill press, bench vise, combination square, surface plate, surface gage, scribe, prick punch, center punch, ball peen hammer, layout ink, hand hack saw, 5/16" drill bit, counter sink, radius gage, 10" flat double cut bastard file, 8" flat single cut smooth file, 3/8-16-NC tap and tap wrench.

PROCEDURE:

1. Select stock as specified.
2. Cut stock to length with power hack saw.
3. Hold work in the bench vise and remove burrs with a file.

   Note: Use soft vise jaws to prevent marring the work.

4. Square one end of the work piece to layout from.
5. Apply layout ink to the work piece.

6. Using the squared end as a reference point locate the center of the 1/16" radius and prick punch.

7. Set the dividers at 1/16" and scribe the arc.

   Note: Layout both sides of the material.

8. Scribe a line $2\frac{1}{4}$" from the end as per drawing.

9. Scribe a line for the angular sides as per drawing.

10. Locate and scribe a line $1\frac{1}{4}$" from the end on all four corners as per drawing.

11. Locate and scribe the 3/32" points on the end on all four sides as per drawing.

12. Scribe the diagional lines from the $1\frac{1}{4}$" marks to the 3/32" marks as per drawing.

13. Locate the center of the 5/16" hole.

14. Center punch the point where the lines intersect.

15. Grip work in the vise for sawing.

16. Saw off the angular side of the peen with the hand hack saw.

   Note: Leave at least 1/32" of material along the line to preserve layout.

17. Rough the angular sides of the peen to the lines with a 10" bastard double cut file.

18. Rough file the radius on the peen end with a 10" bastard double cut file.

   Note: Check the radius with a radius gage.

19. Rough file chamfers on the face end with a 10" bastard double cut file.

   Note: Check angle of chamfers with a combination square.

20. Drill the 5/16" hole.

21. Countersink hole on both sides 1/16".

22. Tap the hole 3/8-16 NC.

23. Finish file all over with a 8" smooth cut file.

24. Polish with emery cloth.

25. Inspect as per drawing.

VOCATIONAL MACHINE SHOP

NAME OF PART TINNERS HAMMER

PROJECT NO. 6 QTY REQ'D 1

MAT'L CRS HT C.H SF

TOLERANCE UNLESS OTHERWISE SPECIFIED

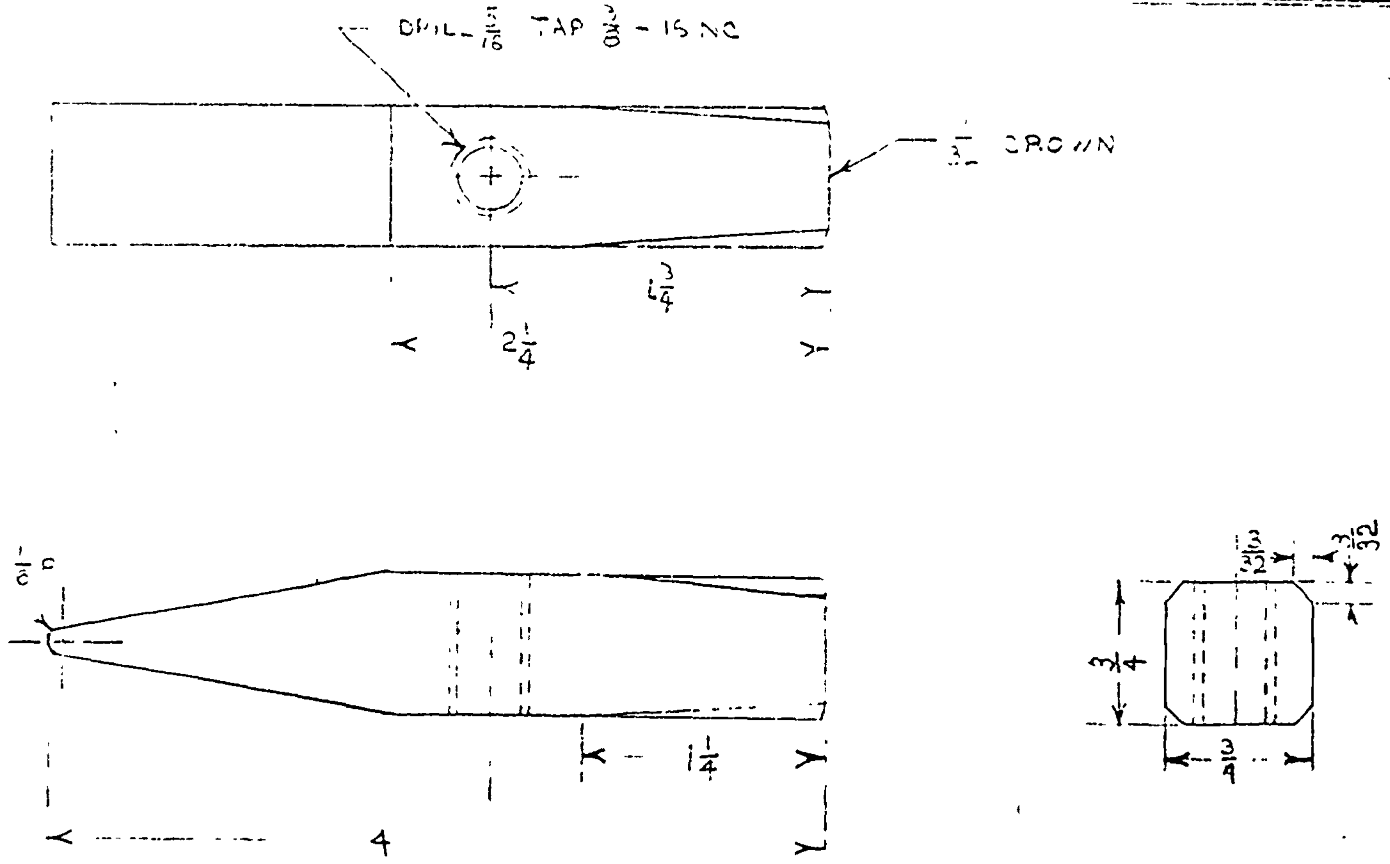
DEC. FRAC $\pm\frac{1}{64}$ ANG.

SCALE FULL SHEET 1 OF 1

DR BY H.S DATE 4-2?-73

NOTE

CHANGE



JOB SHEET

TITLE: TO MAKE A STEP GAGE

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills of straight turning, shoulder truning and knurling.

INFORMATION: Straight turning, shoulder turning and knurling are basic operationsthat a machinist must know how to perform.

SPECIFICATIONS:

STEP GAGE

MATERIAL: Cold Rolled Steel 1 1/8" diameter x 6 5/8" long.

TOOLS AND EQUIPMENT: Power hacksaw, engine lathe, 3 jaw chuck, facing tool, right hand turning tool, tool holder, center drill, drill chuck, 1 ¼" lathe dog, dead center, sleve for dead center, live center, drive plate, steel rule, hermaphrodite calipers, cut off tool, knurling tool, micrometer and abrasive cloth.

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hack saw.
3. Mount work on lathe in 3 jaw chuck.
4. Face end square and center drill.

5. Reverse the work in the lathe.

6. Face the end square and to correct length. BEST COPY AVAILABLE

7. Center drill.

8. Mount work between centers on the lathe.

9. Rough turn end for handle for at least 2 13/16".

10. Finish turn to 3/4" diameter.

11. Undercut 5/8" diameter with cutoff tool for 5/16" long.

12. Knurl with either medium or course diamond.

    Note: Use plenty of oil.
    Set correct speed and feed for knurling.

13. Reverse stock between centers and use 7/8" lathe dog.

    Note: Use brass under srew in order not to mar knurl.

14. Rough turn large 1" diameter.

15. Finish turn 1" diameter with no morethan .002 or .003 to polish with abrasive cloth.

16. Layout ¼" length with hermaphrodite calipers.

17. Rough turn 7/8" diameter.

18. Finish turn 7/8" diameter with no more than .002 or .003 to polish with abrasive cloth.

19. Repeat step 16,17 and 18 for the 3/4", 5/8" 1/2", 3/8" and 1/4" diameters.

20. Cut ¼" excess length off with hand hacksaw.

21. Face end to correct length.

22. Break all sharp edges will mill smooth file.

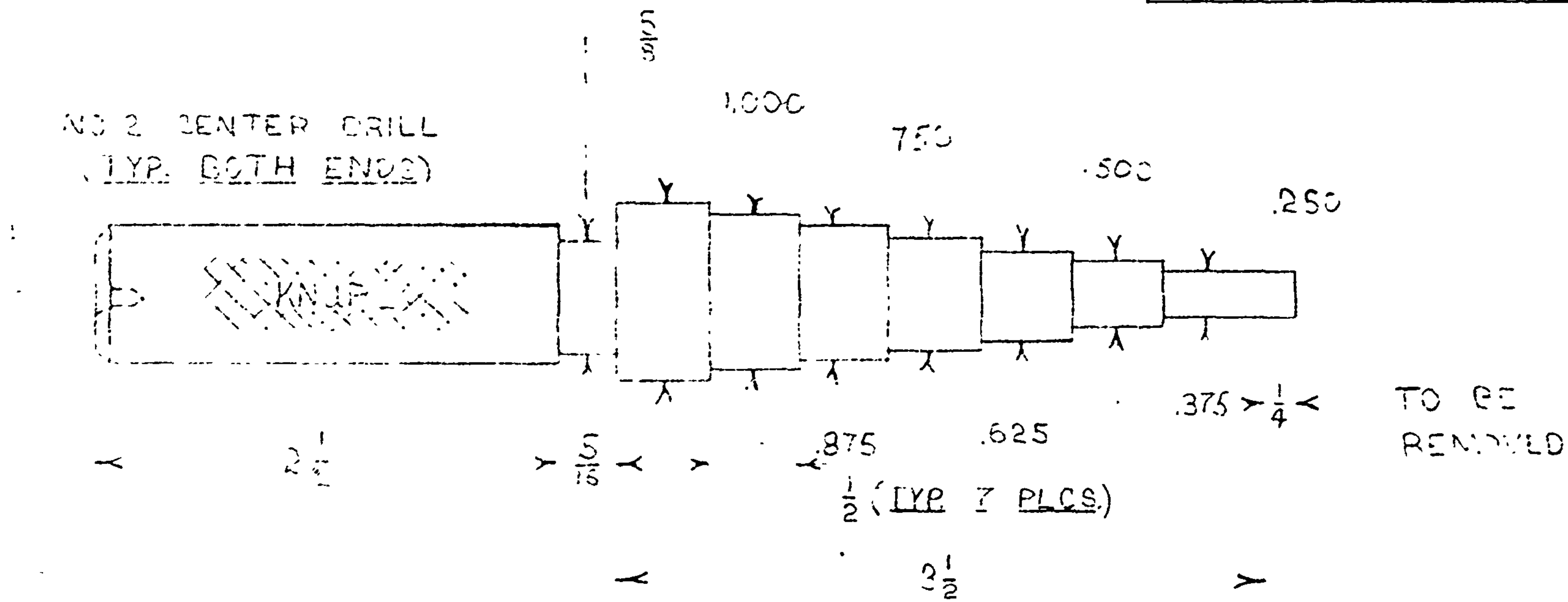
23. Cut chamfer on handle with lathe tool.

24. Polish with abrasive cloth.

25. Inspect as per drawing.

VOCATIONAL MACHINE SHOP

NAME OF PART STEP
PROJECT NO. 3 QTY REQ'D 1
MAT'L CPS H.T S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.005 FRAC. ±1/64 ANG.
SCALE FULL SHEET 1 OF 1
DR BY H.G. DATE
NOTE
CHANGE



BEST COPY AVAILABLE

JOB SHEET

TITLE: TO MAKE A HAMMER HANDLE

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skill in straight or taper turning on the lathe.

INFORMATION: Straight and tapered turning are basic operations that a machinist must know how to perform.

SPECIFICATIONS:

HAMMER HANDLE

MATERIAL: Cold Rolled Steel 7/8" diameter x 9 7/8" long.

TOOLS AND EQUIPMENT: Power hacksaw, steel rule, engine lathe, 3 jaw chuck, facing tool, right hand turning tool, tool holder, center drill, drill chuck, 7/8" lathe dog, dead center, sleeve, live center, drive plate, knurling tool, cut off tool, 31/64" drill bit, 1/2" reamer, 3/8"-16-NC die, die stock and micrometer.

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Mount work on lathe in 3 jaw chuck.

   Note: check lathe chuck to insure it is running concentric.
4. Face end square and centerdrill.
5. Reverse work in the lathe.
6. Face end to correct length and center drill.
7. Mount work between centers.
8. Make at least a 6" roughing cut.

   Note: Check for taper

9. Finish turn to 3/4" diameter.

10. Knurl handle for at least 5½". **BEST COPY AVAILABLE**

Note: Use plenty of oil
Set correct speed and feed for knurling.

11. Cut knurled end to 0.503 diameter ½" long.

12. Reverse stock in lathe and use brass under screw.

13. Rough turn end to 3/8" diameter 3/4" long.

14. Finish turn 3/8" diameter.

15. Set lathe up to cut taper

Note: 1. Use either the tailstock offset method or the taper attachment.

2. Make caculations accurately.

16. Make roughing cut for a distance of at least 4" with 3/8" small diameter and 5/8" large diameter.

17. Check taper carefully to insure proper taper is being cut.

18. Finish turn taper.

19. Cut champher at end of knurl where the taper and knurl join.

20. Place knurled end in 3 jaw chuck protected with brass or in a collet.

21. Cut off 1" of handle with cut off tool.

22. Face end of handle, center drill, drill with 31/64 drill 3 3/4 deep and ream 1" deep with ½" reamer.

23. Face end of cap and cut 45 degree chamfer.

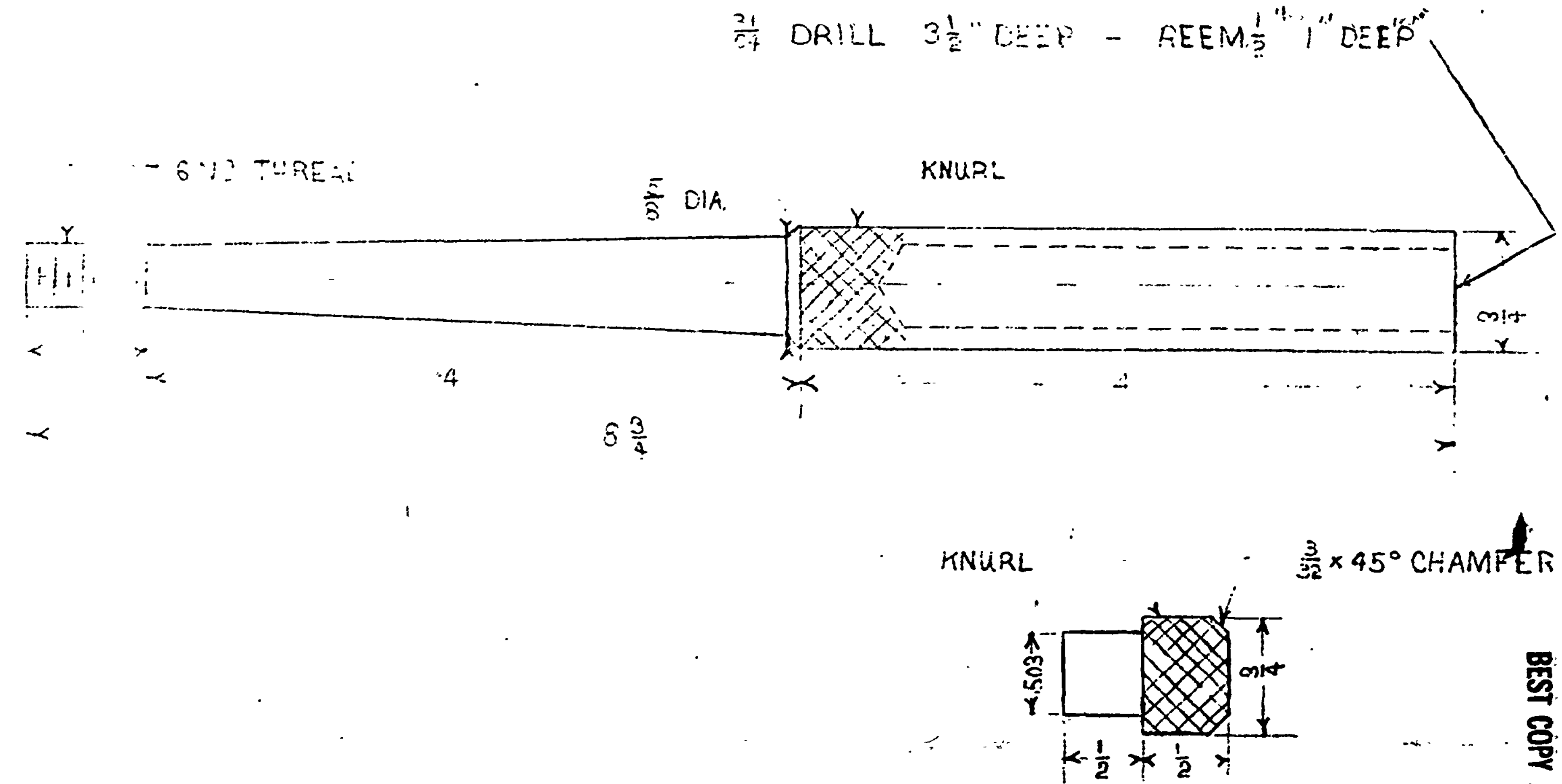
24. Cut 3/8-16-NC thread with die and die stock.

Note: Use plenty of oil.

25. [illegible] cap into end of handle.

26. [illegible]

VOCATIONAL MACHINE SHOP
NAME OF PART
PROJECT NO. QTY. REQ'D
MAT'L
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. FRAC.
SCALE
DR BY DATE
NOTE
CHANGE



JOB SHEET

BEST COPY AVAILABLE

TITLE: TO MAKE A PLUMB BOB

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of the lathe.

INFORMATION: The plumb bob is used in construction or building trades to determine if the item is square with the ground or floor.

SPECIFICATIONS:

PLUMB BOB

MATERIAL: Cold Rolled steel 3/4" Hex stock x $4\frac{1}{4}$".

TOOLS AND EQUIPMENT: Power hack saw, 3/4" hex collect, 3-jaw chuck, engine lathe, lathe tool, 3/8" radius form tool, lay out dye, center drill, 3/32" drill bit, 3/16" drill bit, drill chuck drill press vise, drill press, 8" mill smooth file, hermaphrodite calipers, micrometer.

PROCEDURE:

1. Select stock as specified.
2. Cut stock to length with power hacksaw.
3. Face end square in lathe.
4. Center drill end of stock.

   Note: Use No. 2 Center drill.
5. Drill 3/32" diameter hole 11/64" deep.
6. Apply lay out dye.
7. Scribe a line 3/8" from end with hermaphrodites.
8. Machine to $\frac{1}{4}$" diameter with 3/8" radius form tool back 3/8".
9. Scribe a line 1" from end with hermaphrodites.

10. Machine to 5/8" diameter with 3/8" radius form tool back 1" from end.

11. Scribe a line 2 11/16" from end with hermaphrodites.

12. Turn stock around in the lathe.

13. Set compound rest to cut correct taper.

14. Machine taper.

15. Layout per drawing for 3/16" drilled hole.

16. Center punch hole.

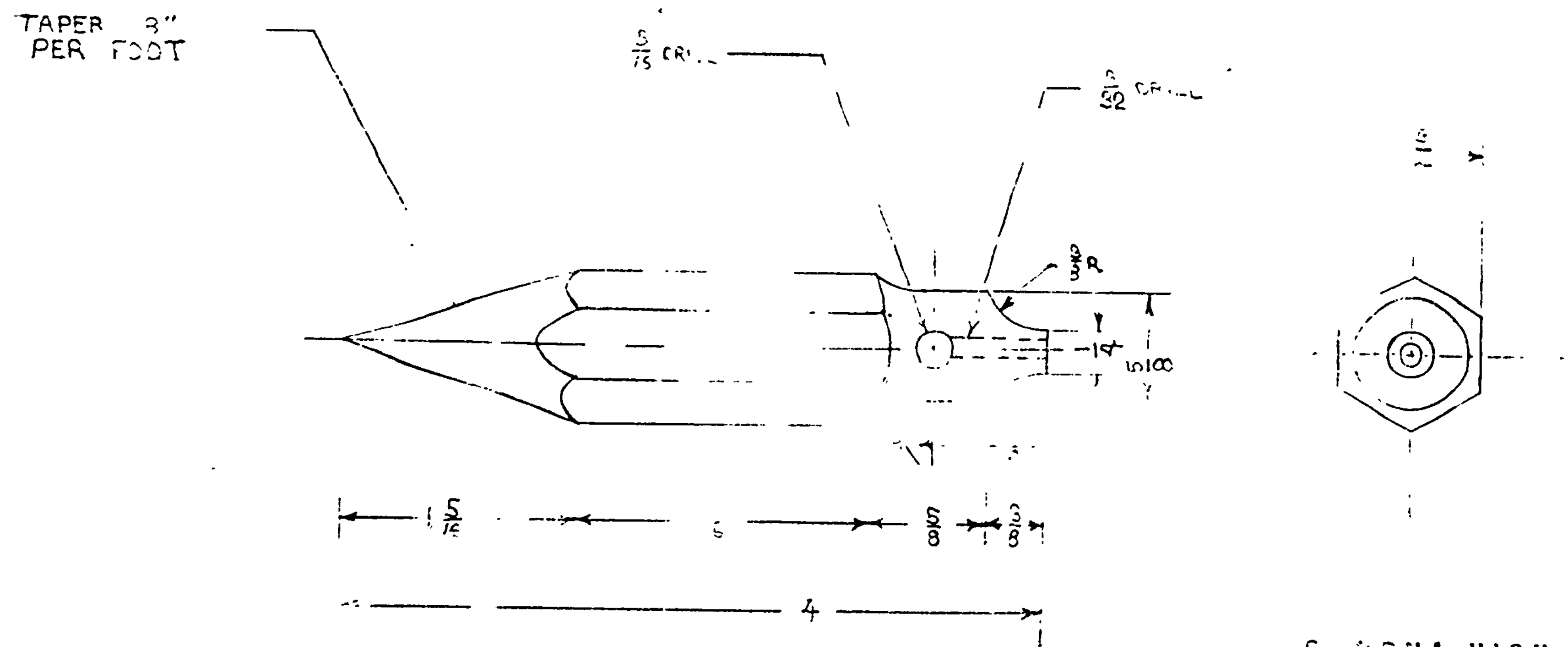
17. Holdstock in drill vise.

18. Center drill and drill 3/16" diameter hole.

19. Polish in lathe to suit.

20. Draw file finish all flat surfaces.

21. Inspect as per drawing.



S... NA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
NAME OF PART PLUMB BOB
PROJECT No. ______ QTY. REQ'D 1
MAT'L C.R.S. H.T. ______ S.F. ______
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ______ FRAC. ± 1/64 ANG. ______
SCALE FULL SHEET 1 OF ______
DR. BY H. G. DATE ______
NOTE ______
CHANGE ______

BEST COPY AVAILABLE

JOB SHEET

TITLE: TO MAKE A V-BLOCK

UNIT: MILL WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of the more common types of bench and layout tools and the use of different types of cutters on the milling machine.

INFORMATION: V-Blocks are very useful tools for machine shop work. They are used for drilling holes, holding round material in the mill and shaper vises, and for bench and assembly work.

SPECIFICATIONS:

V-Block

MATERIAL : Cold Rolled Steel 1¼" x 1¼" x 3 7/8".

TOOLS AND EQUIPMENT: Power hacksaw, combination square set, surface plate, surface gage, layout dye, scriber, ball peen hammer, center,punch, 8" mill smooth file, bevel protractor, Horizontal mill, vertical mill, 1/16" mill saw, ¼" end mill or ¼" horizontal cutter, 90 degree form cutter.

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hack saw.
3. Grip work in the bench vise and remove all burrs with a file.
4. Square end of stock either in mill or shaper.
5. Square second end of stock.
6. Machine excess stock in equal amounts off of the four sides of the block if other than 1¼" [illegible] off [illegible] shaper.

7. Apply layout dye to work piece.

8. Lay out the two 90 degree V's as per drawing.

9. Layout the two 1/16" x 1/16" fillet releases.

10. Layout the two ¼" x 3/32" clamp slots.

11. Machine the two 90 degree V's as per drawing.

    Note: Remove all burrs after each machining operation with a mill smooth file.

12. Machine the 1/16" fillets.

13. Machine the ¼" x 3/32" clamp slots.

14. Inspect as per drawing.

15. Saw block into two equal pieces.

16. Square sawed end and machine to correct length.

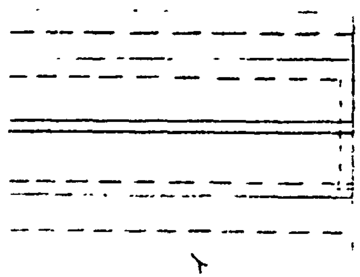
17. Mill 3" x 5/8" x .030" slot on one end of each block.

18. Remove all burrs.

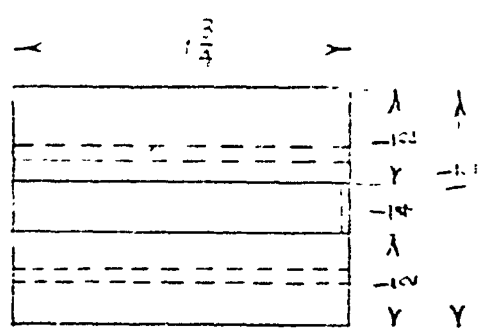
19. Draw file and polish.

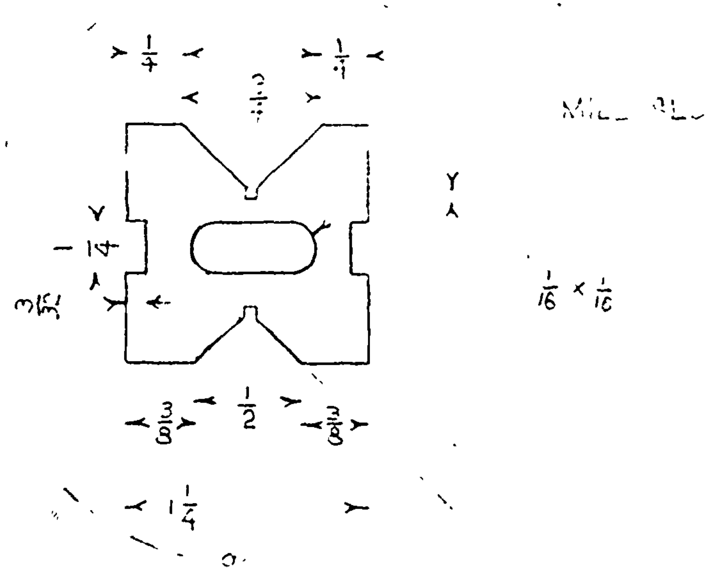
VOCATIONAL MACHINE SHOP

NAME OF PART V-BLOCK
PROJECT NO. 6 QTY REQ'D 2
MAT'L C.R.S. H.T. C.H. S.F. ____
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ____ FRAC. ± 1/64 A.G. ± 1/2
SCALE FULL SHEET 1 OF 1
DR BY ____ DATE ____
NOTE ____
CHANGE ____



CASE HARDEN





BEST COPY AVAILABLE

JOB SHEET

TITLE: TO MAKE A V-BLOCK CLAMP

UNIT: MILLING MACHINE AND LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of various machines and sequence of operations.

INFORMATION: V-Block clamps are used with the V-Block to hold work in V-Block while various machining operations or bench and assembly work is being done.

SPECIFICATIONS:

V-BLOCK CLAMP

MATERIAL: Cold Rolled Steel ½" x 1 5/8" x 2 7/16"

TOOLS AND EQUIPMENT: Power hack saw, milling machine, layout dye, scribe, combination square, spring dividers, prick punch, ball peen hammer, drill press, center drill, appropriate drills, band saw, 10" flat double cut bastard file, 8" flat mill smooth file, ½"-28-NF tap and tap handle, engine lathe, 3 jaw chuck, facing tool, right hand turning tool, tool holder, drill chuck, steel rule, hermaphrodite calipers, threading tool, knurling tool, micrometer.

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hack saw.
3. Square end and machine to length on mill.
4. Apply lay out dye.
5. Locate center of 5/8" radius and prick punch.
6. Set spring dividers at 5/8" radius and scribe arc.
7. Set spring dividers at 13/16" radius and scribe

Note: Layout both sides of material.

BEST COPY AVAILABLE

8. Scribe [illegible] for top of clamp ¼" each side of center line.

9. Scribe a [illegible] wide from center line of 5/[illegible] clamp.

10. Scribe a [illegible] over from edge of clamp at [illegible] slot guide.

11. Scribe a [illegible] up from bottom of clamp.

12. Locate [illegible] of hole to be taped and center [illegible].

13. Center [illegible] to be taped.

14. [illegible] taped with no. 3 drill and counter [illegible].

15. [illegible]

Note: [illegible] cutting oil while tapping [illegible]

16. [illegible] radius.

17. [illegible] drill, 1" drill and [illegible]

Note: [illegible] has [illegible] put through [illegible] set up in mill and [illegible] for better results.

18. Saw [illegible]

Note: [illegible] material along the [illegible] preserve layout.

19. [illegible] portion with a bastard file, [illegible] lines.

Note: [illegible]

20. [illegible] with a [illegible] smooth cut file.

21. [illegible]

22. [illegible]

MATERIAL: Cold rolled steel [illegible] diameter x 3" long.

PROCEDURE:

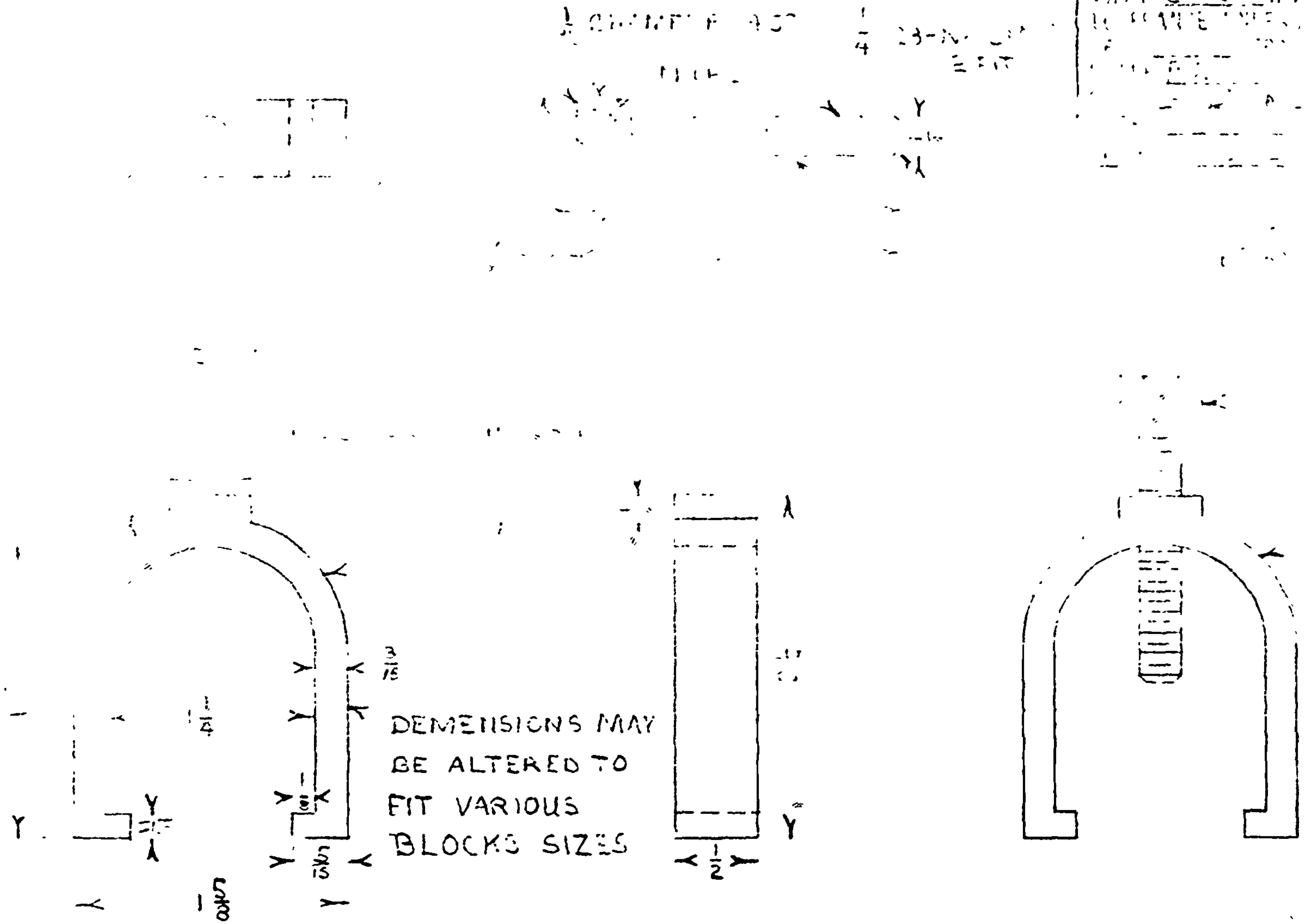
1. [illegible] specified.
2. [illegible] power hacksaw.
3. Mount stock on lathe in 3 jaw chuck.
4. Face [illegible].
5. [illegible] in the lathe.
6. [illegible] and center drill.
7. Knurl [illegible] for 1".
8. [illegible] and grip 3/4".

   Note: [illegible] with brass strip.
9. [illegible] for 1 7/8".
10. [illegible] ear and slow down R.P.M.
11. [illegible] threading tool to machine [illegible]
12. [illegible] fit with taped hole.
13. [illegible] threads with mill smooth [illegible]
14. [illegible] threaded end and cut 1/32x [illegible]
15. [illegible] end.

HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART V-BLOCK CLAMP

PROJECT NO. 7 QTY. REQ'D 2



JOB SHEET

TITLE: TO MAKE A "C" CLAMP

UNIT: LATHE AND MILLING MACHINE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of layout, mill and lathe work and the sequence of operations necessary to make part.

INFORMATION: A "C" clamp is an all purpose clamp made in the shape of a C. It is in general used for all kinds of work.

SPECIFICATIONS:

"C" CLAMP

MATERIAL: FRAME - Cold Rolled steel 5/8"x2"x3 5/8" long.
Handle - Cold Rolled Steel 3/16" diameter x 2¼" long.
Swivel - Cold Rolled Steel 5/8" diameter x 2¼" long.
Screw - Cold Rolled Steel ½" diameter x 4 1/8" long.

TOOLS AND EQUIPMENT: Combination square, scribe, spring dividers, layout dye, center drill, appropriate drills, proper size end mills, radius gages, 10" bastard file, 8" mill smooth file, drill press vise, drill press, drill chuck, engine lathe, tool holder, turning tool, threading tool, lathe dog, drive plate, V-Block and clamp; surface plate and surface gage.

PROCEDURE:

FRAME

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Set up work in shaper vise and square end.
4. Reverse work and square other end.
5. Set square in vise and shape to correct length
6. Apply layout dye.
7. [illegible]

Note: Use surface plate and surface gage.

8. Locate center of each radius inside "C"clamp, center punch and scribe radius with spring dividers.

9. Centerdrill each center of radius.

10. Drill ¼" radius with 31/64" drill and 1/8" radius with 7/64" drill.

11. With band saw cut out center section of frame.

12. Place stock in vise of milling machine.

    Note: Check vise for squarness.

13. Machine inside of frame.

14. Finish file the inside with 8" mill smooth file. Use a round smooth filefor fillets in corners.

15. Layout and centerpunch hole to be drilled for screw.

16. Layout radius on corners and ends per drawing.

17. Layout angle on anvil end of frame.

18. Mount work in vise on vertical mill.

    Note: Set up square in vise with anvil end in bottom of vise.

19. Centerdrill and drill hole with 5/16" drill and countersink.

20. Tap with tape started indrill chuck with 3/8-16-NC tap.

21. File radius with 10" flat bastard file.

22. Finish file radius with 8" mill smooth file.

23. Mount work in shaper vise and cut angle on anvil end.

24. File all surfaces smooth with 8" mill smooth file.

    Note: Break all sharp edges.

25. Inspect per drawing.

HANDLE

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Place stock in a 3 jaw chuck on lathe with 1" projecting.
4. Round end with form tool.
5. Turn stock end for end with 1" projecting.
6. Round end as in step 4.
7. Polish with emery cloth.

SWIVEL

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Place stock in 3-jaw chuck, projecting 1".

   Note: Check to insure work is running concentric.
4. Set proper R. P. M.
5. Face end of stock square.
6. Rough turn large diameter to 21/32" in diameter.
7. Center drill and drill with ¼" drill.
8. Set compound rest at an angle of 35 degrees to the right.
9. Cut angle.
10. Set cutoff tool properly.
11. Cut off part 5/16" long.
12. Inspect as per drawing.

SCREW

PROCEDURE:

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck and face end square.

4. Centerdrill end of stock.

5. Turn stock end for end in chuck.

6. Face stock to correct length and centerdrill.

7. Rough turn diameter of section to be threaded.

8. Turn stock end for end on centers.

9. Rough turn diameter of head section.

10. Finish turn diameter of head section.

11. Turn stock end for end on centers.

    Note: Use brass under lathe dog to protect finish.

12. Finish turn diameter of section to bethreaded.

    Note: Turn diameter .005" undersize to insure a free fit.

13. Face sholder of head section.

14. Set proper speeds and feeds for threading.

15. Set up threading tool properly.

16. Cut thread to fit tapped hole in the frame.

17. Rough turn small diameter or pilot section.

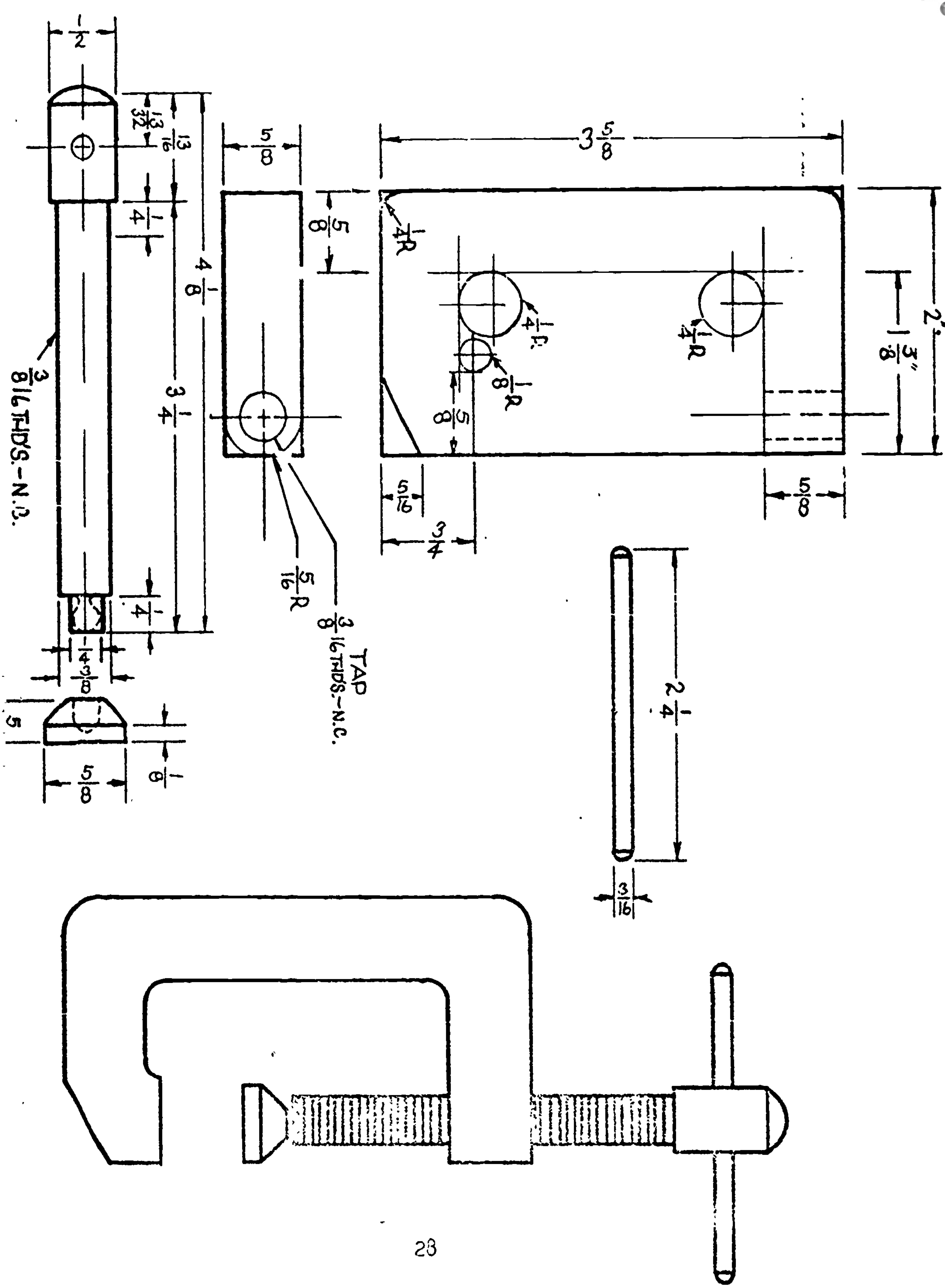
18. Use form tool to cut ball on end of screw.

19. Use form tool to cut radius on head of screw.

20. Place in V-Block and layout hole as per drawing.

21. Center drill. drill and ream 3/16" diameter hole in head of screw.

22. Inspect as per drawing.



MYRNA HIGH SCHOOL MACHINE SHOP	C - CLAMP FULL SCALE" PRO. NO. 16	DRN. BY: H. G. CKD. BY: RES

JOB SHEET

TITLE: TO MAKE A 60° LATHE CENTER

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in taper turning on the lathe by the tailstock offset method, taper attachment and the compound rest.

INFORMATION: A soft lathe center is used in the headstock of the lathe when turning stock between centers.

REFERENCE:

SPECIFICATIONS:

60° LATHE CENTER

MATERIAL: Cold Rolled steel 1 1/16" diameter x 5 1/8" long.

TOOLS AND EQUIPMENT: Power hacksaw, engine lathe, 3-jaw chuck, facing tool, right hand turning tool, tool holder, center drill, drill chuck, lathe centers, lathe dog, layout dye, drive plate, steel rule, hermaphrodite calipers, micrometer, taper gage, 10" lathe file, 8" mill smoothe file,

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Mount work in 3-jaw chuck.
4. Face the end square and center drill.
5. Reverse the work in the lathe.
6. Face the end square and center drill.
7. Mount work between centers on the lathe.
8. Arrange lathe for taper turning.

   Note: Use taper attachment or the tail stock offset method. Check caculations carefully.

9. Take a light trim cut over tapered section.

10. Test the taper in a No. 3 Morse Taper gage.

    Note: If taper is not correct, adjust setting to correct error.

11. Take second cut and test as before. Repeat until you have taper correct.

12. Rough turn taper, allowing .030" stock for finishing cut.

13. Finish turn taper allowing .003" - .004" for filing.

    Note: Make 3 or 4 marks the length of the taper with chalk. Insert taper into No. 3 Morse Taper Gage and turn clockwise only and check forhigh spots on taper.

14. File high spots to fit gage properly.

15. Polish lightly with fine emery cloth and oil.

16. Undercut end for clearance at small end of taper as per drawing.

    Note: Break all sharp corners with mill smooth file.

    Note: When taper shank is inserted in taper gage there should be a space of 5/8" between end of gage and beginning of 60° angle on center point.

17. Remove face plate and center from headstock spindle.

18. Place tapered shank in headstock spindle.

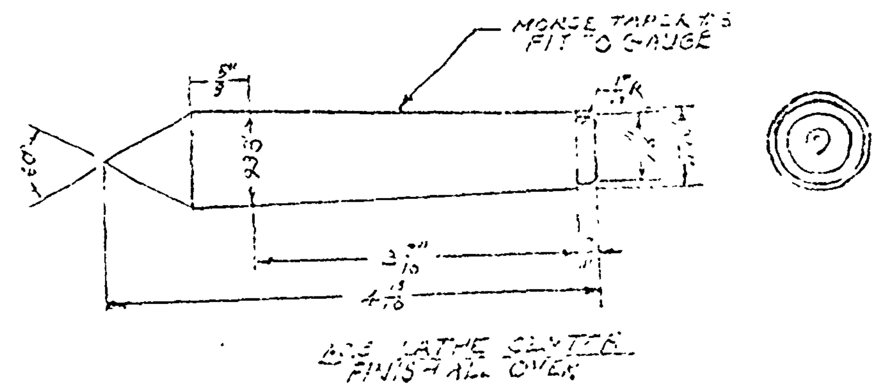
19. Set compound rest at an angle of 60 degrees.

20. Turn the angle on cneter point which is 30 degrees from center.

    Note: Check angle of point with thread center gage.

21. Inspect as per drawing.

STOCK - $1\frac{1}{2}$ D x $4\frac{15}{16}$" LONG T.S.



SMYRNA HIGH SCHOOL

VOCATIONAL MACHINE SHOP

NAME OF PART: LATHE CENTER
PROJECT No. 9 QTY. REQ'D 1
MAT'L C.R.S. H.T. S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. FRAC. ± 1/64" ANG. ± 1/2°
SCALE 1/2 SHEET OF 1
DR. BY C.D. DATE 7-2-74
NOTE
CHANGE

BEST COPY AVAILABLE

JOB SHEET

TITLE: TO MAKE A MACHINIST'S CLAMP

UNIT: SHAPER AND LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: TO Develop skills in the use of different machines, use of handand layout tools and the sequence of operations.

INFORMATION: A machinist clamp may be adjusted to fit a piece of work by means of a screw passing through the center of each jaw. Another screw in the end of one jaw is used to put pressure on the other jaw. It is used by machinist for holding small parts at the bench and at machines.

SPECIFICATIONS:

MACHINIST'S CLAMP

MATERIAL: Part No. 1 and 2. Cold rolled steel 5/8" x 5/8" x 4 13/16" long.
Part no. 3 and 4. Cold Rolled steel 11/16 diameter x 3 7/8" long.
Part No. 5. Cold Rolled steel 3/16" diameter x 2 9/16" long.

TOOLS AND EQUIPMENT: Power hacksaw, engine lathe, cutting tool, tool holder, 3 jaw check, 4-jaw chuck, combination square, scribe, layout dye, shaper, 10" baster file, 8" mill smooth file, centerpunch, ball peen hammer, drill press vise, drill press, drill chuck, center drill, countersink, F drill, 21/64" drill, threading tool and V block, lathe dog and drive plate.

PROCEDURE:

Parts 1 and 2 JAWS

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Place stock in 4-jaw chuck with 1" projecting.

Note: Adjust jaws so stock runs true.

4. Set proper spindle speed.

5. Face one end only.

6. Transfer to work bench and lay out bevel at faced end of stock.

7. Mount work in shaper and shape beveled section.

8. Layout 5/16" radius on ends.

9. File radius and check with radius gage.

10. Layout and centerpunch holes to be drilled in jaw No. 1.

11. Place jaw No. 1 on top of jaw No. 2 in position they will occupy when assembled and clamp together.

12. Drill hole with F drill at beveled end, drilling through both pieces.

13. Drill hole with F drill at opposite end through jaw No. 1 and part way into jaw No. 2 as per drawing.

14. Redrill hole through jaw No. 2 at beveled end with 21/64" drill.

15. Square bottom of blind hole in jaw No. 2 using a drill ground flat on bottom.

16. Tap 5/16-18-NC in jaw No. 1.

17. Countersink tapped hole in jaw No. 1 to depth of ½ thread.

18. Break sharp edges with a file.

19. Draw file and polish all surfaces.

20. Inspect per drawing.

Parts 3 and 4 SCREWS

PROCEDURE:

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. [illegible]

Note: Check to insure chuck is running concentric.

BEST COPY AVAILABLE

4. Face end and centerdrill

5. Reverse work in lathe.

6. Face end to correct length and centerdrill

7. Place between centers on lathe and drive with lathe dog.

8. Set proper spindle speed.

9. Apply layout dye.

10. Layout as per drawing.

11. Rough turn section to be threaded.

12. Turn stock end for end on centers.

13. Rough turn large diameter to rough size for shoulder.

14. Rough turn recessed section, using small round nose tool ground to form fillets at shoulders.

15. Finish turn large diameter.

16. Finish turn recessed section.

17. Set form facing tool and form radius on head end.

18. File radius on collar as per drawing.

19. File all machined surfaces lightly to remove tool marks and also break all sharp edges.

20. Polished finished section.

Note: Use fine emery cloth and oil.

21. Turn work end for end on centers.

Note: Use brass under lathe dog to protect finish.

22. Finish turn section to be threaded.

Note: Turn .005" undersize to insure a freet fit.

23. [illegible] screw with round [illegible].

24. Set proper speeds and feeds for threading.

25. Set up threading tool properly.

26. Cut thread to fit tapped hole in jaw No. 1.

27. Round end of screw No. 3 with forming tool.

28. Turn small section at end of thread on screw No. 4 for pilot.

    Note: This should be a slip fit in blind hole in jaw No. 2.

29. Center punch for drilled hole through head section.

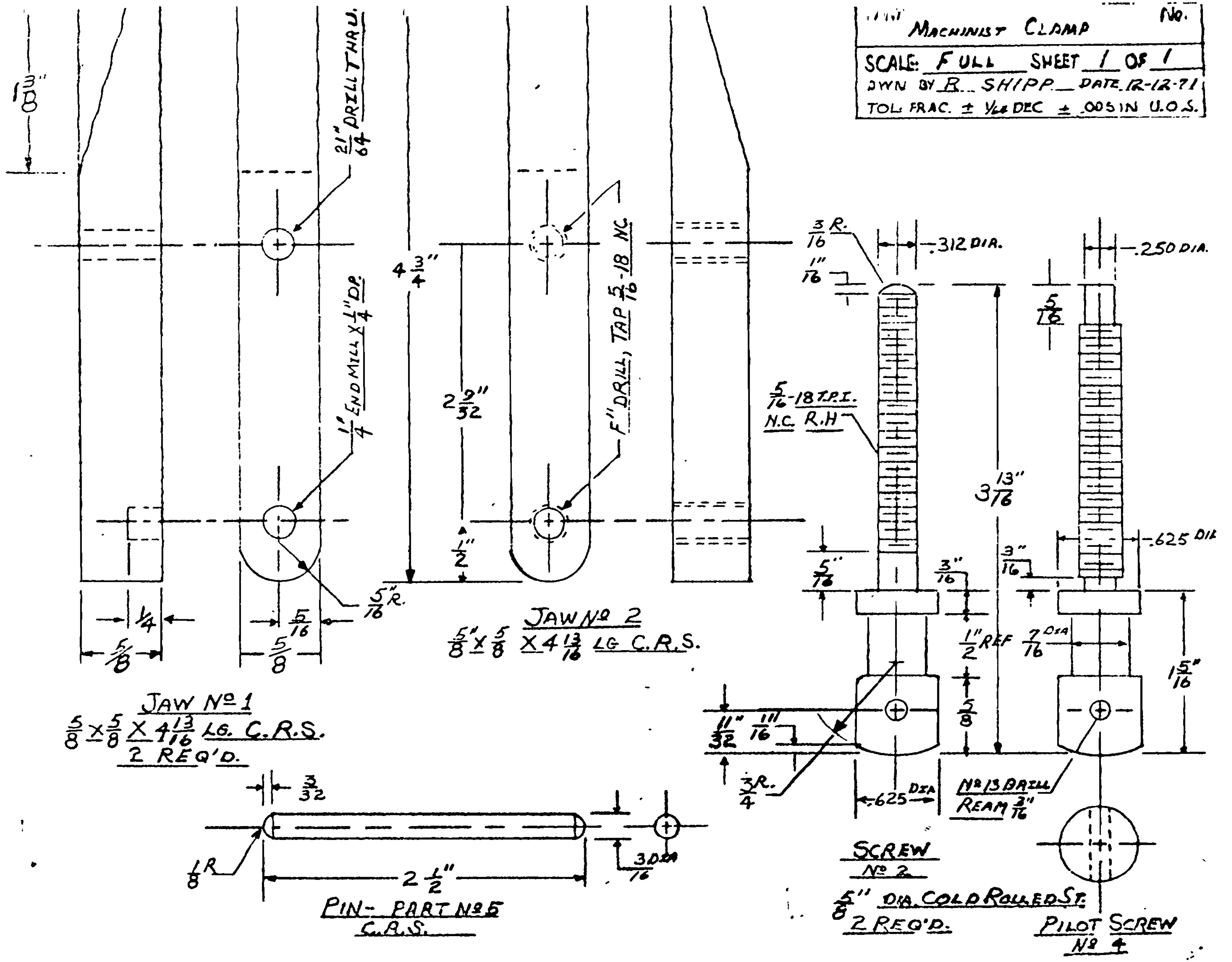
30. Drill hole through head section for drive fit of Part No. 5.

31. Inspect as per drawing.

Part No. 5 PIN

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Place work in 3-jaw chuck with 1" projecting.
4. Round end with form tool.
5. Turn stock end for end with 1" projecting.
6. Round end as in step 4.
7. Assemble as per drawing.
8. Inspect as per drawing.


MACHINIST CLAMP
No.
SCALE: FULL SHEET 1 OF 1
DWN BY R. SHIPP DATE 12-12-71
TOL: FRAC. ± 1/64 DEC ± .005 IN U.O.S.
JAW No 1
5/8 x 5/8 X 4 13/16 LG. C.R.S.
2 REQ'D.
21/64" DRILL THRU.
1/4" END MILL X 1/4" DP.
5/16" R.
JAW No 2
5/8" x 5/8 X 4 13/16 LG C.R.S.
F" DRILL, TAP 5/16-18 NC
4 3/4"
2 9/32"
1/2"
1 3/8"
PIN - PART No 5
C.R.S.
2 1/2"
3/32
1/8 R
3/16 DIA
SCREW
No 2
5/8" DIA. COLD ROLLED ST.
2 REQ'D.
.312 DIA.
3/16 R.
1/16"
5/16-18 T.P.I.
N.C. R.H
3 13/16"
5/16"
3/16"
1/2" REF
5/8
11/32"
11/16
3/4 R.
.625 DIA
No 13 DRILL
REAM 3/16"
PILOT SCREW
No 4
.250 DIA.
.625 DIA
7/16 DIA
1 5/16"

JOB SHEET

TITLE: CLAMP LATHE DOG

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in the use of layout tools and various machines and sequence of operations.

INFORMATION: The clamp lathe dog is used to drive square and rectangular work pieces between centers on the lathe. The tail is inserted into the drive plate.

SPECIFICATIONS:

CLAMP LATHE DOG

MATERIAL: Cold Rolled Steel 7/8" x 7/8" x 4 7/16" long-Jaw
Cold Rolled Steel 7/8" x 7/8" x 7 5/16" long-Tail Jaw
Cold Rolled Steel 13/16 Diameter x 4 13/16 long-Screw

TOOLS AND EQUIPMENT: Power hacksaw, layout dye, scribe, combination square, milling machine, center punch, ball peen hammer, 10" double cut bastart file, 8" mill smooth file, drill press vise, drill press, 15/32" drill, engine lathe, centers for lathe, lathe dog, facing tool, right hand turning tool, tool holder, knurling tool, threading tool, 3/8" drill, 7/16-14-NC tap, and index head.

PROCEDURE:

PLAIN JAW

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Layout and mark location of 90 degree notch in center of stock.

4. Locate and draw a line through the center lengthwise of the stock to intersect the lines already drawn for location of 90 degree notch.

5. Starting from this center, layout and mark location of holes to be drilled. Layout radius on ends of stock.

6. Mount work in vise of vertical mill and machine the 90 degree notch.

7. Drill 15/32" clearance holes 2 places for screws.

8. Rough file radius with 10" bastard file.

   Note: check radius with radius gage.

9. Finish file radius with 8" mill smooth file.

10. Draw file all surfaces.

   Note: Break all sharp edges and corners.

11. Inspect as per drawing.

PROCEDURE:

TAIL JAW

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 4-Jaw chuck

   Note: Check to insure work is running concentric.

4. Set lathe for proper R. P. M.

5. Face end of stock.

6. Reverse stock in lathe and face end to correct length.

7. Set taper attachment to cut taper of 1 9/16" per foot.

   Note: Taper may be cut with tailstock offset method.

8. Take trial cut on tapered section.

   Note: Check for correct taper.

9. Rough turn tapered section.

10. Finish turn tapered section.

11. File tapered section lightly to remove machine marks.

12. Polish with a fine emery cloth and oil.

13. Layout center of notch.

14. Layout radius on end of jaw.

15. Mount work in vise on vertical mill and machine 90 degree notch.

16. Rough file radius with 10" bastard file.

17. Finish file radius with 8" mill smooth file.

18. Clamp jaw on top of tail jaw in position they will occupy when assembled. Be sure the 90 degree notches are opposite each other.

19. Through the plain jaw spot centers of holes in tail jaw with 15/32" drill.

20. Drill two 3/8" holes in tail jaw.

21. Tap two holes 7/16-14-NC in tail jaw.

    Note: Use plenty of cutting oil.

22. Draw file all surfaces.

    Note: Break all sharp edges and corners.

23. Heat with acetylene torch and bend tail as per drawing.

24. Inspect as per drawing.

PROCEDURE:

SCREW

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck.

4. Face end square and center drill.

5. Reverse work in lathe.

6. Face end to correct length and center drill.

7. Place stock between centers on lathe and hold with bent tail lathe dog.

8. Rough turn small diameter section.

9. Reverse stock in lathe.

10. Rough turn large diameter section.

11. Finish turn large diameter section.

12. Set lathe for knurling use medium knurl.

13. Knurl all of large diameter.

    Note: Use plenty of oil.

14. Rough turn head section.

15. Finish turn head section.

16. Rough end of head section with form tool.

17. Reverse stock in lathe.

    Note: Protect stock with brass under lathe dog

18. Finish turn small diameter.

    Note: Turn small diameter .005" undersize to insure free fit.

19. Face sholder of knurled section.

20. Set lathe for proper speeds and feeds for threading.

21. Set threading tool properly.

22. Cut thread to fit tapped hole in tail jaw.

23. Round end of threaded section with form tool.

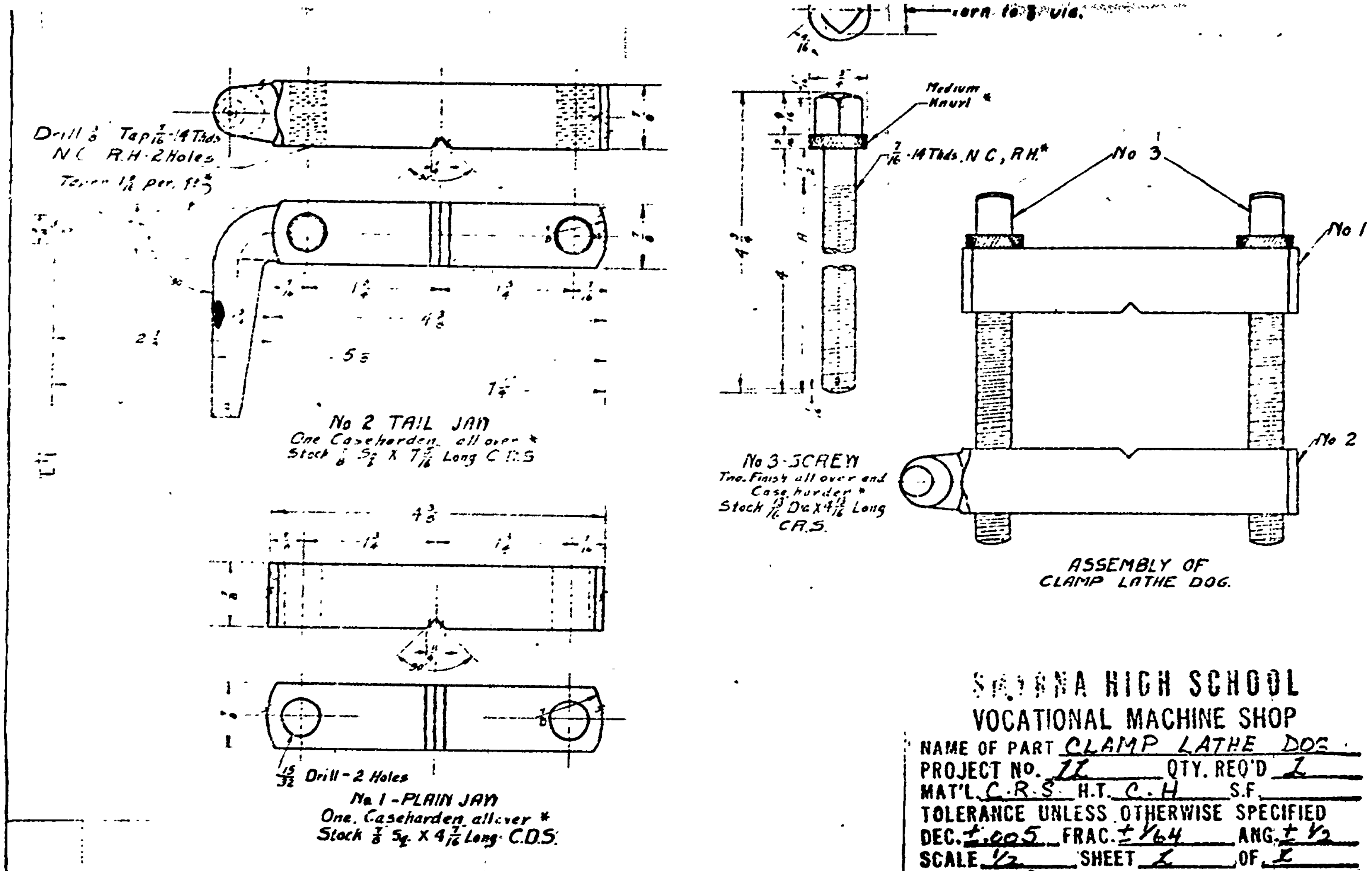
24. Mount work in index head on vertical mill.

25. When end of cutter touches work raise table .004" and cut flat.

26. Index 10 turns and cut flat.

27. Repeat step 27 for next two flats.

28. Inspect as per drawing.



S[illegible]NA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART CLAMP LATHE DOG
PROJECT NO. 11 QTY. REQ'D 1
MAT'L C.R.S H.T. C.H S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.005 FRAC. ±1/64 ANG. ±1/2
SCALE 1/2 SHEET 1 OF 1
DR. BY C.D. DATE
NOTE
CHANGE

JOB SHEET

TITLE: TO MAKE A BALL-PEEN HAMMER

UNIT: LATHE AND MILL

OCCUPATION: MACHINIST

OBJECTIVE: To acquaint the student with the use of form tools on the lathe and the use of the index head on the milling machine.

INFORMATION: A small ball peen hammer is used on small layout and very small or light work.

REFERENCE:

SPECIFICATIONS:

BALL-PEEN HAMMER

MATERIAL: Tool steel ½" diameter x 2" long.

TOOLS AND EQUIPMENT: Layout dye, hermaphrodite calipers, engine lathe, facing tool, right hand turning tool form tools, tool holder, 8" mill smooth file, 23/64" drill, #6 taper pin reamer, edge finder, milling machine, index head and mill cutter.

PROCEDURE:

1. Select stock as specified.
2. Cut to length on power hacksaw.
3. Mount work in lathe collect.
4. Face end square.
5. Reverse work in lathe.
6. Face end square and to length.
7. Apply layout dye.
8. Scribe 5/8" line with hermaphrodite calipers for the head.
9. With form tool of 9/64" radius set center of tool on scribed line and plunge cut to .290 diameter.

   Note: Use plenty of oil while plunge cutting.

10. Reverse stock in lathe.

11. Mark center of .235 diameter with hermaphrodite calipers.

12. With form tool of 9/64" radius set center of tool on scribed line and plunge cut to .235 diameter.

    Note: Use plenty of oil while plunge cutting.

13. Machine diameter to .430 diameter for ball end.

14. Machine ball with a 7/64" radius form tool.

15. File crown of $1\frac{1}{4}$" radius on face end.

16. Chuck stock in index head.

17. With edge finder find center of hole to be drilled.

18. Center drill and drill 25/64" diameter.

19. Ream to center from each side with .350 #6 taper pin reamer.

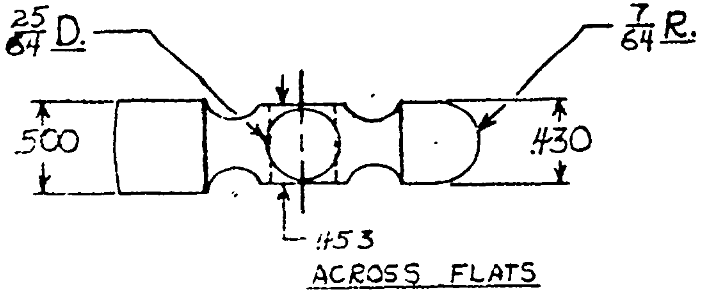
20. Index 10 turns.

21. Cut flat with bottom of end mill to .477 diameter.

22. Index 20 turns and machine second flat to .453 across flats.

23. Polish and break all sharp edges.

24. Inspect as per drawing.



VOCATIONAL MACHINE SHOP

NAME OF PART BALL-PEEN HAMMER

PROJECT NO.____ QTY REQ'D 1

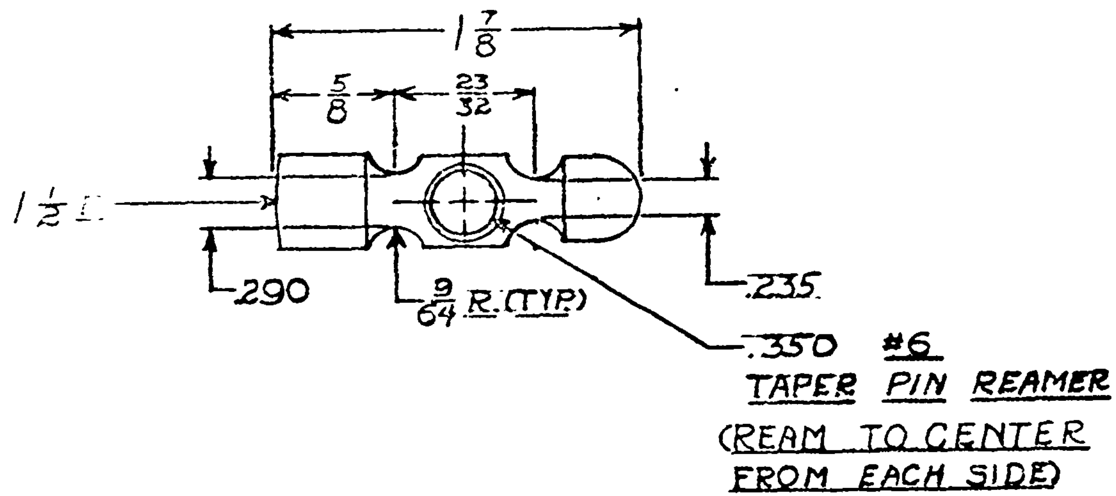
MATERIAL S1 TS H T C-58 SF. POL.

TOLERANCE UNLESS OTHERWISE SPECIFIED-

DEC ± .003 FRAC. ± 1/64 ANG. ____

SCALE 1"=1" DATE 1-25-71

DRAFTING BY Rick Robinson



JOB SHEET

TITLE: HAND CENTER PUNCH

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in straight turning, shoulder turning, taper turning and knurling.

INFORMATION: A hand center punch is used to centerpunch layouts for drilling.

SPECIFICATIONS:

HAND CENTER PUNCH

MATERIAL: MACHINERY STEEL 1 1/8" diameter x 4" long Hammer.

Cold Rolled Steel 3/8" hex x 2¼" long Knob.

Tool steel 9/16" diameter x 5 9/16" long Punch.

TOOLS AND EQUIPMENT: Power hacksaw, engine lathe, 3-jaw chuck, 4-jaw chuck, dead center live center, lathe dog, face plate, facing tool, right hand turning tool, tool holder, knurling tool, form tools, center drill, appropriate drills, taps and dies, layout dye, hermaphrodite calipers, 8" mill smooth file and emery cloth.

PROCEDURE:

HAMMER

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Mount work in 4-jaw chuck projecting two inches. Indicate chuck so as stock runs true.
4. Face end of stock square and center drill.
5. Place live center in tailstck and adjust to support work.
6. Rough turn diameter.
7. Finish turn diameter.
8. [illegible] to obtain a 15 degree taper.
9. Turn taper on nose of work.

Note: Feed tool with compound rest.

10. Finish file all machined surfaces with a 8" mill smooth file to remove all machine marks.

11. Set lathe to knurl large diameter.

    Note: Use plenty of oil.

12. Drill work as per drawing.

13. Cut off work to length as per drawing plus 1/32" to face end.

14. Mount work in lathe chuck with 1/4" of stock projecting and true as before.

15. Face end to length as per drawing.

16. Round edges with a 8" mill smooth file.

17. Polish with emery cloth and oil.

18. Inspect as per drawing.

PROCEDURE:

KNOB

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck with one inch projecting and running true.

4. Face end of stock square.

5. Chamfer corners.

6. Centerdrill end of stock.

7. Drill to size as per drawing.

8. Tap as per drawing.

    Note: Hold tap in drill chuck in tailstock spindle.

    Turn headstock by hand.

9. Cut work off to length as per drawing, plus 1/32" to face end.

10. Mount work in lathe chuck with unfinished end [illegible]

11. Face end of stock square.

12. Face to length as per drawing.

13. Round end and form fillet with forming tools. as per drawing.

14. Polish with emery cloth.

15. Inspect as per drawing.

PROCEDURE:

PUNCH

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck.

4. Face end of stock square and centerdrill.

5. Reverse work in lathe chuck.

6. Face end of stock to length, plus 5/16" to remove the center hole in punch end.

7. Rough turn 1/4" diameter.

8. Rough turn large diameter.

9. Finish turn large diameter.

10. Finish turn 1/4" diameter.

    Note: Allow .003" for polishing to size.

11. Finish turn section to be threaded .005" undersize.

12. Face shoulder of large diameter.

13. File all finished surfaces lightly to remove all machine marks.

14. Remove work from centers and place in lathe chuck.

15. Hold die square with tailstock and die stock resting on tool holder.

16. Keep a steady pressure with tailstock, RPM very slow and use plenty of oil.

17. Cut thread to fit tapped hole in knob.

18. Reverse work in lathe chuck.

19. Rough turn 3/16" diameter.

20. Finish turn 3/16" diameter.

21. Face shoulder of large diameter section to length as per drawing.

22. Face punch end to length.

23. Set compound rest at proper angle to obtain a taper of 11½ degrees.

    Note: This is approximately 96 degrees to the right.

24. Turn tapered section

    Note: Feed with compound rest.

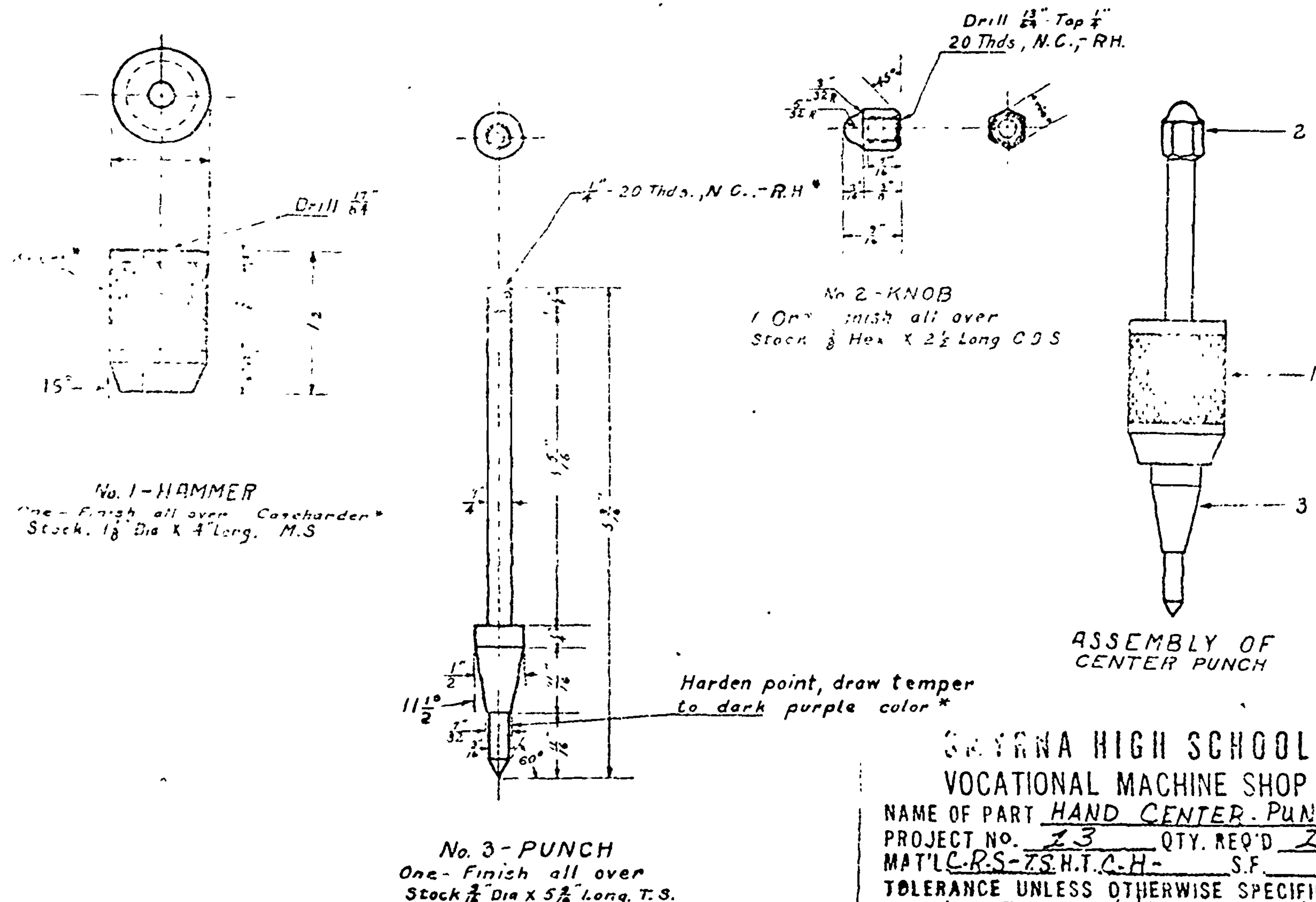
25. Set compound rest at an angle of 120 degrees to the right.

26. Turn angle as per drawing.

27. Polish all machined surfaces lightly to remove machine marks.

28. Hardened and draw temper.

29. Inspect as per drawing.



SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART HAND CENTER PUNCH
PROJECT NO. 23 QTY. REQ'D 1
MAT'L C.R.S-T.S. H.T. C.H- S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.005 FRAC. ±1/64 ANG. ±1/2
SCALE HALF SHEET 1 OF 1
DR. BY H. G. DATE 11-9-73
NOTE T.S. DRAW TO DARK PURPLE
CHANGE

JOB SHEET

TITLE: HOW TO MAKE PARALLELS

UNIT: SHAPER WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in operating the shaper, layout, use of drill press and surface grinder after hardening.

INFORMATION: Parallels are used with thin stock so that it may be raised to a convenient height in the vise to machine or do work on.

SPECIFICATIONS:

PARALLELS

MATERIAL: Cold Rolled Steel per chart.

TOOLS AND EQUIPMENT: Shaper, drill press, surface grinder, steel rule, 8" mill smooth file, layout dye, scriber, center punch, ball peen hammer, combination square, shaper tool, 1" micrometer, 2" micrometer, center drill, drills as listed on chart.

PROCEDURE:

1. Select stock as specified per chart.
2. Cut to length with power hacksaw.
3. Square end and cut to length on vertical mill.
4. Layout per chart for shaper.
5. Shape to drawing size.

   Note: Leave .007" - .008" for grind stock per side.
6. Layout for drilling.
7. Drill holes as per drawing size on chart.

JOB SHEET

8. Countersink all holes.

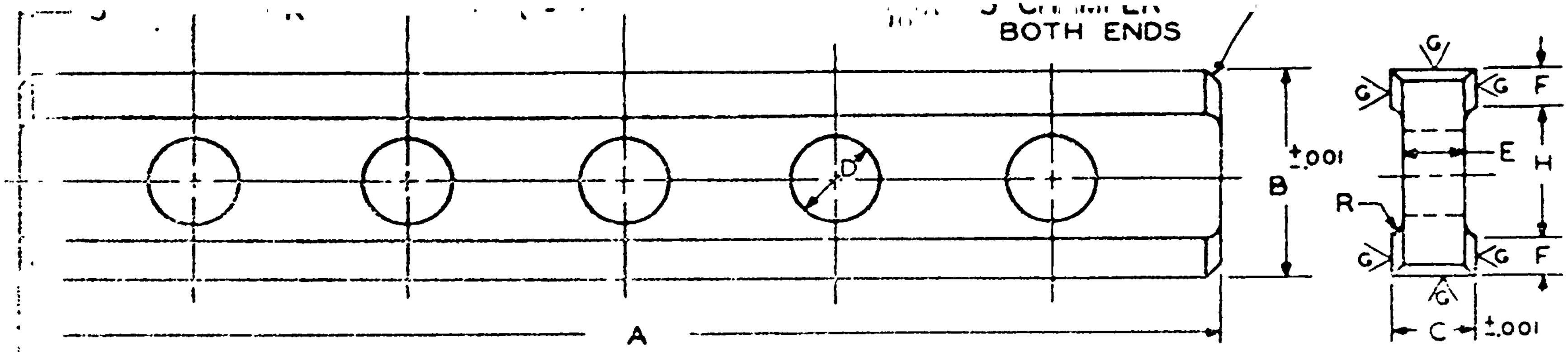
   Note: Use correct size countersink to deburr holes.

9. Break all sharpe edges.

10. Inspect as per drawing.

11. Harden and draw temper.

12. Grind to chart size.



[illegible] NO.	STOCK			OTHER DIMENSIONS						
	A	B	C	D	E	F	H	J	K	R
1	4	1/2	3/8	——	1/4	1/8	1/4	——	——	1/16
2	4 1/2	5/8	3/8	——	1/4	3/16	1/4	——	——	1/16
3	5	3/4	3/8	——	1/4	3/16	3/8	——	——	1/16
4	5 1/2	7/8	3/8	——	1/4	3/16	1/2	——	——	1/16
5	6	1	1/2	——	3/8	1/4	1/2	——	——	1/16
6	6 1/2	1 1/8	1/2	3/8	3/8	1/4	5/8	1	1 1/8	1/16
7	7	1 1/4	1/2	1/2	3/8	1/4	3/4	1	1 1/4	1/16
8	8	1 1/2	3/4	5/8	1/2	5/16	7/8	1 1/4	1 3/8	1/8

NOTE:

ALLOW .010, EACH SURFACE, FOR GRINDING

SMYRNA HIGH SCHOOL

VOCATIONAL MACHINE SHOP

NAME OF PART PARALLELS
PROJECT No. 14 QTY. REQ'D 1
MAT'L. C.R.S. H.T. C.H. S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.005 FRAC. ± 1/64 ANG. ± 1/2
SCALE SHEET 1 OF 1
DR. BY H. G. DATE 3-27-73
NOTE
CHANGE

JOB SHEET

TITLE: HOW TO MAKE HOLD-DOWNS

UNIT: SHAPER WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in use of the shaper. To develop skills using hold downs in a shaper on parallels and angular shaping.

INFORMATION: Hold downs are used in holding thin stock above the top of the vise in order that machining operations may be performed. They are wedge shaped in cross section with the thick edge beveled 203 degrees. This causes the hold-down to press downward at the thin edge when brought against the work.

SPECIFICATIONS: HOLD-DOWNS

MATERIAL: Cold Rolled Steel 3/8" x 1" x 6 1/8"

TOOLS AND EQUIPMENT: Power hacksaw, shaper, surface grinder, verticle mill, milling cutter, combination square, scribe, layout dye, gage blocks, sine plate.

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Set work in shaper vise.
4. Shape one side with the use of hold-downs.
5. Shape opposite side as per drawing.

   Note: Leave .015" over all grind stock.
6. Rough out third side to 15/16".
7. Shape fourth side to 7/8" plus grind stock.

   Note: Leave .015" over all grind stock.

8. Mill ends square to finished length.

9. Shape out 1/32" x 5/32" section.

10. Shape angle on top as per drawing.

11. Harden and draw temper.

12. Dress grinding wheel.

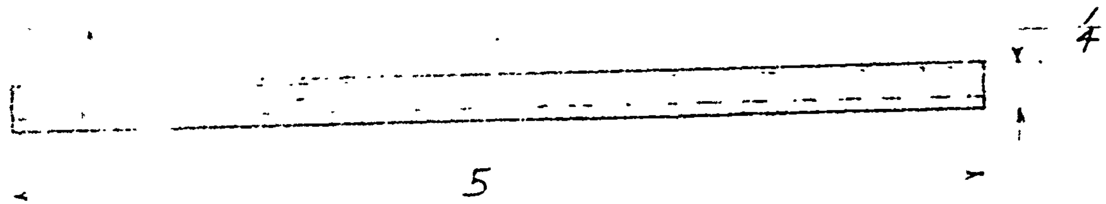
13. Finish grind both sides of 5/16" dimension.

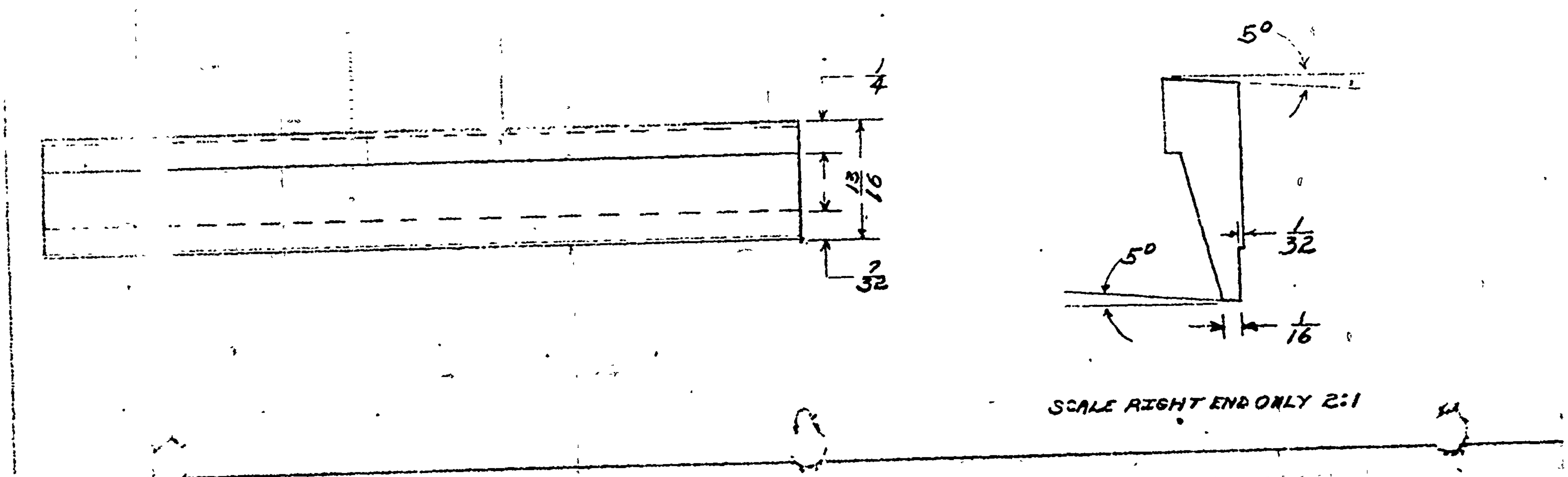
14. Grind 6 degree angle and opposite side as per drawing.

15. Inspect as per drawing.

VOCATIONAL MACHINE

NAME OF PART HOLD DOWNS
REQ'D 2 EA
MATERIAL C.1045 HT C.H.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.005 FRAC ± 1/64 ANG ±3
SCALE 1:1 SHEET 1 OF 1
DR. BY R. SHIPP DATE 12-25
NOTE LEAVE GRINDING STOCK
CHANGE





JOB SHEET

TITLE: TO MAKE A BENCH BLOCK

UNIT: LATHE AND DRILL PRESS WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in bench layout, lathe and drill press operations.

INFORMATION: The bench block is very useful for drill press work, knocking out pins and bench assembly.

SPECIFICATIONS:

BENCH BLOCK

MATERIAL: Cold Rolled Steel 3 1/8" diameter x 1 5/8" long.

TOOLS AND EQUIPMENT: Power hacksaw, engine lathe, 3-jaw chuck, drill press, steel rule, center drill, facing tool, right hand turning tool, knurling tool, 8" mill smooth file, layout dye, combination square, dividers, center punch, ball peen hammer, appropriate drills and reamers, vertical mill and milling cutter.

PROCEDURE:

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Mount work in lathe and face end square.
4. Center drill.
5. Drill 9/16" pilot hole.
6. Drill [illegible] through 9/16" hole.
7. Ream hole to [illegible]

   Note: [illegible] lathe. [illegible] oil.

8. [illegible] diameter 1" deep.
9. [illegible] 15 degree angle as [illegible]

BEST COPY AVAILABLE

10. Place on lathe [illegible].

11. Mount work [illegible] 3-[illegible] chuck.

    Note: Check to [illegible] chuck in running contcertric.

12. Rough turn [illegible] diameter.

13. Finish turn [illegible] diameter.

14. Face opposite en[illegible] to [illegible] thickness.

15. Knurl outsi[illegible].

    Note: Use medium knurl.
    Use plenty of oil.

16. Break all sharp edges with 8" mill smooth file.

17. Apply layout dye

18. Layout as per [illegible].

19. Centerdrill all [illegible].

20. Drill all [illegible] per drawing.

21. Ream all holes [illegible] per drawing.

22. Set work in [illegible] head on vertical mill.

23. Tilt [illegible] 45 degree angle.

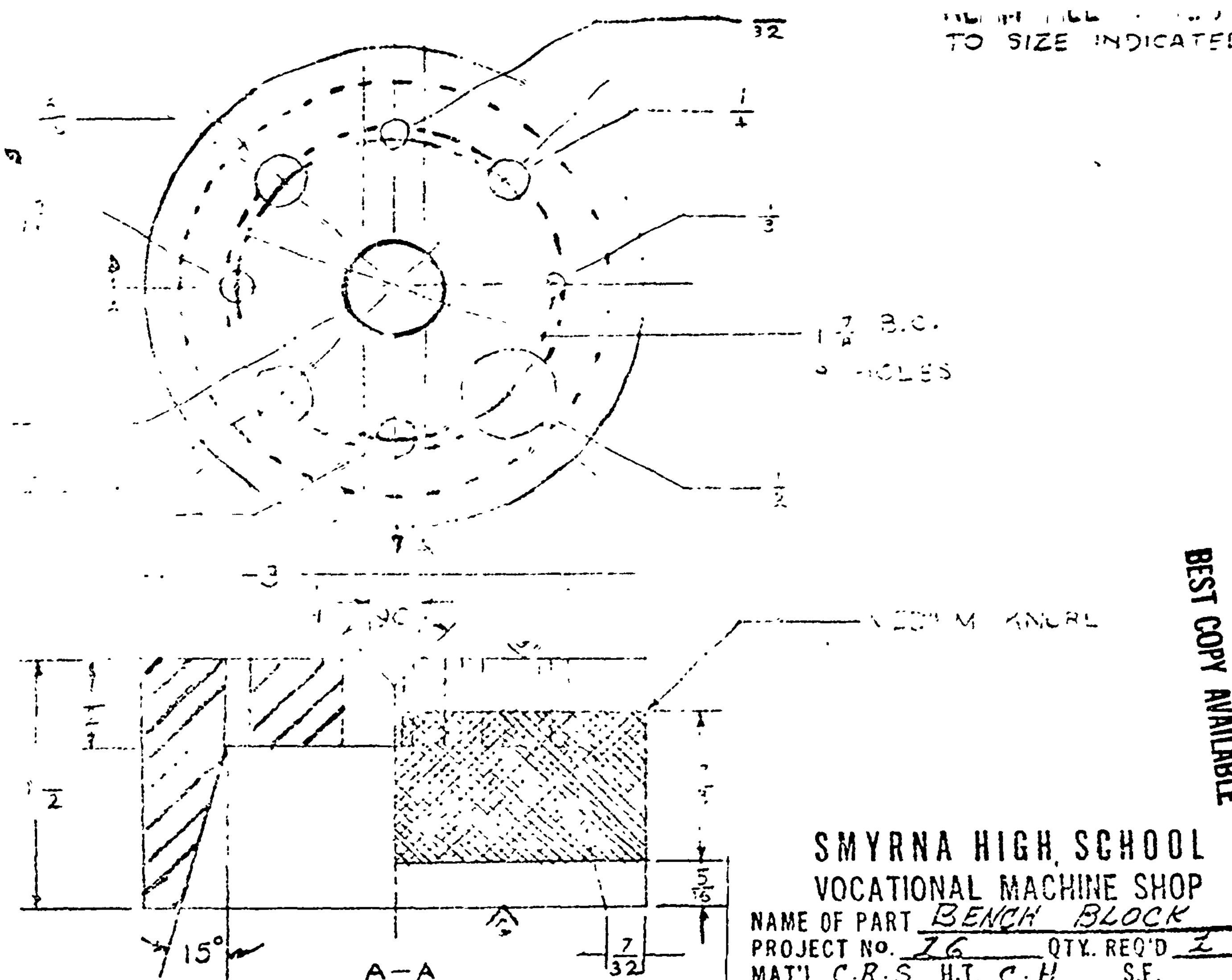
24. Break all [illegible].

25. Polish all [illegible].

26. Inspect [illegible].

27. Harden [illegible].

28. Surface [illegible].



SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART BENCH BLOCK
PROJECT No. 26 QTY. REQ'D 1
MAT'L C.R.S H.T. C.H S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.005 FRAC. ±1/64 ANG. ±1/2
SCALE FULL SHEET 1 OF 1
DR. BY H.G. DATE 2-17-73
NOTE
CHANGE

JOB SHEET

BEST COPY AVAILABLE

TITLE: "T" HANDLE [illegible]

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop [illegible] internal, external taper turning and [illegible] on the horizontal mill, using the index head.

INFORMATION: A T-handle tap wrench is used to hold and turn small taps [illegible] 1/2" in diameter. The chuck or nut when tightened holds the tap securely. It is also useful for turning small hand reamers.

SPECIFICATIONS:

"T" HANDLE TAP WRENCH

MATERIAL: Nut - Cold rolled steel 1 1/16" diameter x [illegible].

Handle - Cold rolled steel 5/16" diameter x 5 5/1[illegible] long.

Stem - Cold rolled steel 7/8" diameter x 3 7/8" long.

TOOLS AND EQUIPMENT: [illegible] engine lathe, facing tool, right [illegible], cutoff tool, knurling tool, [illegible], live center, center drill, [illegible] plate, lathe dog, 10" [illegible] file, [illegible] mill smooth file, appro[illegible] mill, milling saw, [illegible] boring bar.

PROCEDURE:

1. [illegible]

2. [illegible]

3. [illegible]

BEST COPY AVAILABLE

4. Face end of stock.

5. Centerdrill and support work with center in tailstock.

6. Rough turn diameter.

7. Finish turn diameter.

8. Rough turn taper.

9. Knurl with either medium or course knurl.

   Note: Use plenty of oil.
   Set correct speed and feed for knurling

10. Finish turn taper.

11. Drill into end of work as per drawing.

12. File tapered section with 8" mill smooth file, break all sharp edges. Polish tapered section with fine emery cloth and oil.

13. Cut off work as per drawing.

    Note: Use plenty of oil on cut off tool.

14. Mount work in 3-jaw chuck, with unfinished end projecting.

15. Face end to length

16. Drill 1 1/8" deep with 5/8" drill.

17. Bore hole to size and depth as per drawing.

18. Bore threading recess.

19. Tap hole 3/4-16-NF.

    Note: Use starting tap, plug tap and then bottom tap.
    Use plenty of oil.

20. Reverse work in chuck.

21. Bore tapered hole.

22. Chamfer [illegible] of thread and outside corner.

23. [illegible]

PROCEDURE:

HANDLE

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Place stock in lathe check.
4. Face and round ends to length.
5. Inspect as per drawing.

PROCEDURE:

STEM

1. Select stock as specified
2. Cut to lenth with power hacksaw
3. Mount work in 3-jaw chuck.
4. Face end of stock.
5. Reverse work in chuck.
6. Face end of stock to length.
7. Rough turn all diameters.
8. Finish turn all diameters except the tapered section.

   Note: Turn the threaded diameter .005" undersize for free fit.

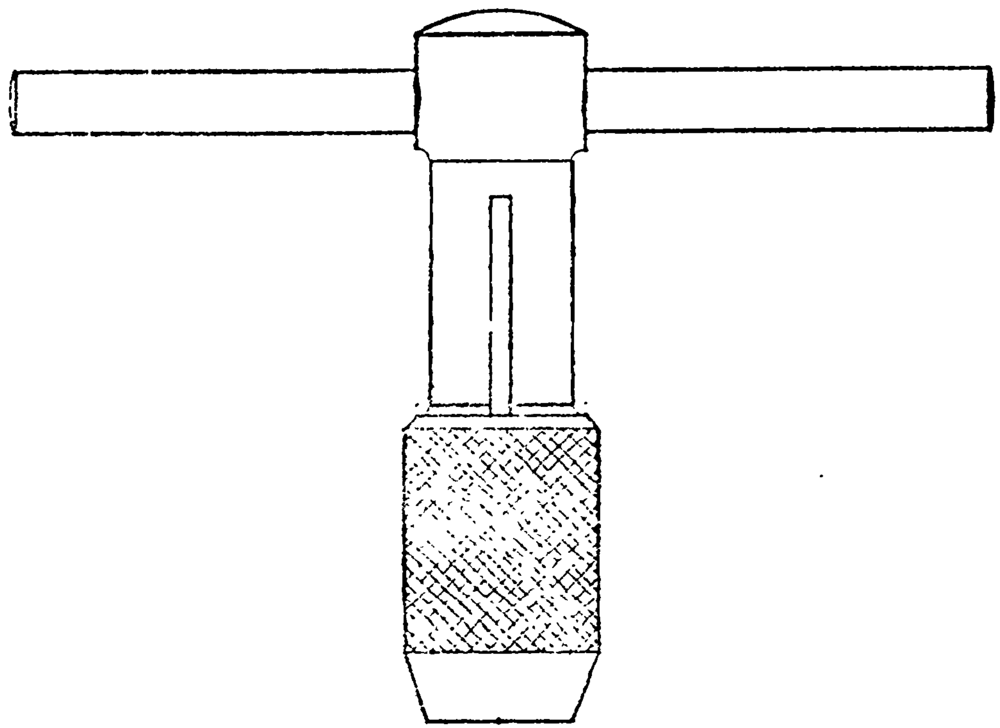
9. Finish turn all fillets.
10. Set proper seed and feed for threading.
11. Cut thread to fit tapped hole in nut.
12. Drill hole in end of stem as per drawing.
13. Layout, center punch, drill and ream hole through diameter of head.
14. Mount work in index head on horizontal mill.
15. Center work over cutter.
16. Set proper speed and feed on mill.

17. Mill one slot [illegible] over halfway through stock.

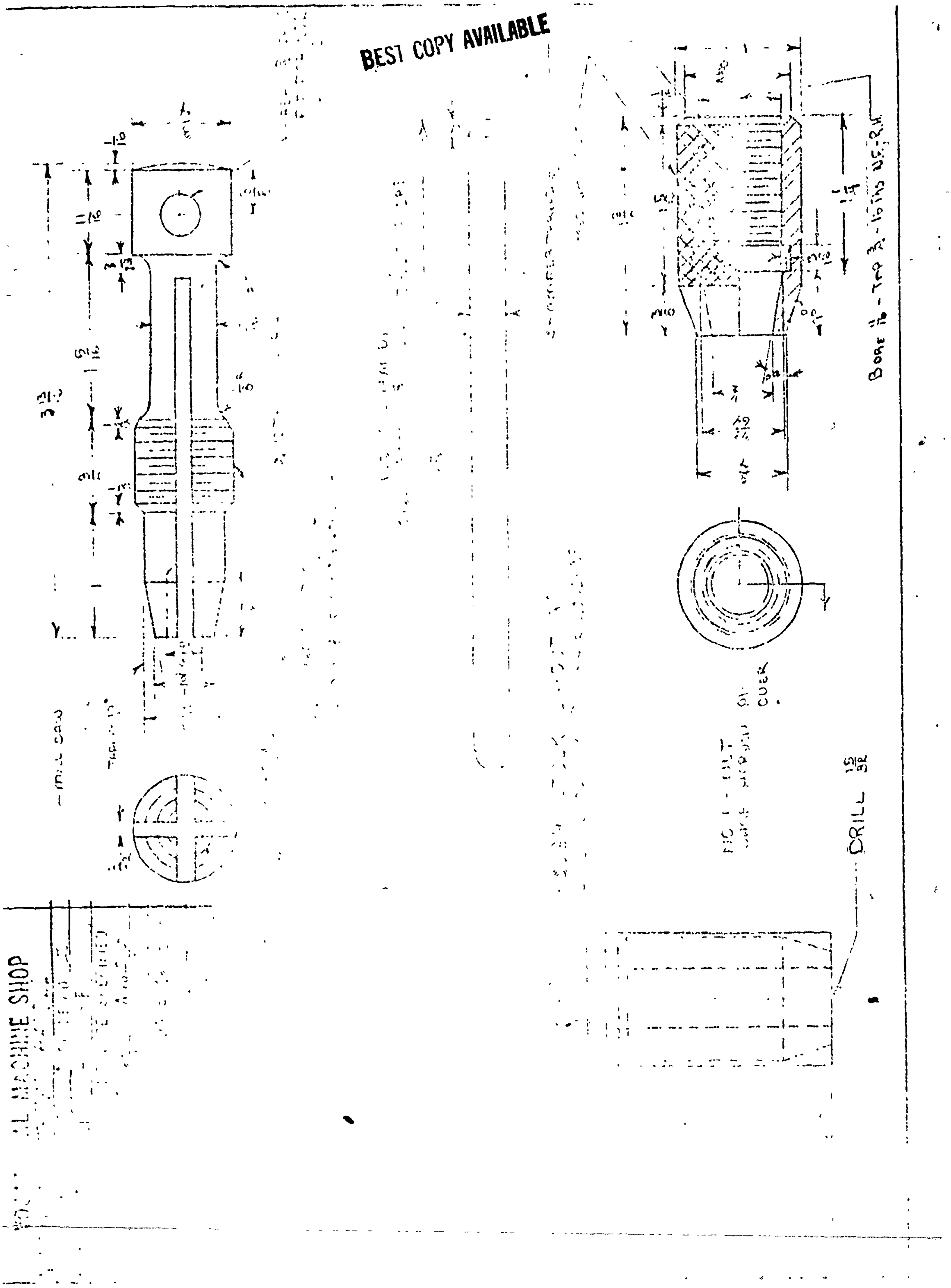
BEST COPY AVAILABLE

Note: 10 turns of the handle indexes part 90 degrees.

18. Inspect as per drawing.



SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
NAME OF PART T-TAP HANDLE
PRO. KIT NO. ______ [illegible]
MAT'L ______ H.T ______
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ______ FRAC. ______ ANG. ______
SCALE ______ SHEET ______ OF ______
DR. BY H.G ______ DATE ______
NOTE ______
CHANGE ______


MACHINE SHOP
DRILL 15/32
COVER

JOB SHEET

TITLE: ADJUSTABLE TAP WRENCH

UNIT: LATHE WORK AND MILL WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in straight turning, threading, knurling and assembly of parts after machining.

INFORMATION: An adjustable tap wrench is a straight type wrench having a V-shaped opening in the center. A sliding member, or adjustable jaw operated by one of the handles makes it possible to hold taps of various sizes. This type wrench is made in many sizes to turn taps and reamers of all sizes.

SPECIFICATIONS:

ADJUSTABLE TAP WRENCH

MATERIAL: Adjusting sleeve - Cold Rolled steel 11/16" diameter x 5" long.

Body - Cold Rolled Steel 3/4" diameter x 7 5/16" long.

Plunger - Tool Steel ¾" diameter x 4 3/16" long.

Spring - Music wire .047 diameter x 2 1/8" long.

TOOLS AND EQUIPMENT: Power hacksaw, engine lathe, vertical mill, facing tool, right hand turning tool, tool holder, 3-jaw chuck, dead center, live center, lathe dog, drive plate, drill chuck, centerdrill, 10" flat double cut bastard file, 8" mill smooth file, layout dye, centerpunch, scribe, combination square, ball peen hammer, threading tool, appropriate drills, steady rest, reamer and countersink.

PROCEDURE:

A. [illegible]

1. Select [illegible] material.

2. [illegible]

3 Mount work in 3-jaw chuck

4. [illegible]

5. Centerdrill end of stock.

6. Place live center in tailstock and support adjusting sleeve.

7. Rough turn large diameter.

8. Rough turn smaller diameter.

7a. Finish turn smaller diameter.

8a. Finish turn large diameter.

9. Knurl larger diameter with medium or fine diamond.

   Note: Use plenty of oil.
   Set correct speed and feed for knurling.

10. Break all sharp edges with a 8" mill smooth file.

11. Polish small diameter with fine emery cloth and oil.

12. Drill end of stock with tap drill to depth.

13. Tap hole, supporting and guiding tap with tailstock center.

14. Cut to length, plus 1/32" for facing end.

15. Mount work in lathe chuck.

    Note: [illegible] to protect knurl.

16. Face end of sleeve.

17. File radius on end with 10" lathe file.

18. Polish radius end.

19. Layout 1/4" hole in end of sleeve.

20. Center punch, drill and countersink hole.

21. [illegible]

PROCEDURE:

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck.

4. Face end of stock and centerdrill.

5. Reverse stock in the chuck.

6. Face end to length and centerdrill.

7. Mount work between centers on lathe.

8. Rough turn all three diameters.

9. Rough turn fillets at shoulders.

10. Finish turn all three diameters.

11. Finish turn fillets at shoulders.

12. Set up lathe and threading tool to machine 1/2-20-NC.

13. Cut threads and check fit with tapped hole in adjusting sleeve.

14. Set work in vise on milling machine.

15. Mill flat surface on center section.

16. Layout, centerpunch and drill hole through center flat section.

17. Mount work in 3-jaw chuck on lathe. Set up steady rest on 7/16" diameter to support work.

18. Drill and ream hole through handle to meet drilled hole the t____ flat section

19. Saw 1/16" slot through threaded section.

20. Mount work in vise on milling machine with flat section on parallels with drilled hole clear of parallels.

21. Broach 90 degree notch in body to hold tap.

22. File and polish all machined surfaces.

23. Inspect as per drawing.

PROCEDURE:

PLUNGER

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck.

4. Face and square ____________________

5. Reverse work in lathe chuck.

6. Face end square and to lenth, plus 5/8" stock for removing centers.

7. Centerdrill end of stock

8. Mount work between centers on lathe.

9. Rough turn two diameters.

10. Face shoulder to length.

11. Finish turn diameters, leave .003"-.004" stock on small diameter for filing to size.

12. File for slip fit in to reamed hole through handle of body.

13. Place work in lathe chuck with small end projecting and face end to length, removing the center.

14. Reverse work in lathe chuck and face large end to length, removing the center.

15. Turn 30° angle on large diameter by using the compound rest.

16. Set part in vise on milling machine with the side of part at 45 degrees to bottom of vise.

17. Machine 90 degree notch and 45 degree angles on small diameter.

18. Layout centerpunch and drill for 1/16" pin.

19. Hardened all over and draw temper.

20. Inspect as per drawing.

PROCEDURE:

[illegible]

1. Select a [illegible] rod 4" long and drill a hole [illegible] 1¼" from end with a No.66 [illegible].

2. Place rod in [illegible] drilled end projecting [illegible].

3. [illegible] with center [illegible]

4. Run music wire (18 gauge) between two wood or fiber blocks clamped in tool post. Bend a hook on end of wire 1/4" long and at right angles.

5. Place hook through hole in rode (from bottom side) and run cross slide back until wire is tight.

6. Place lathe in back gear at slow R.P.M.

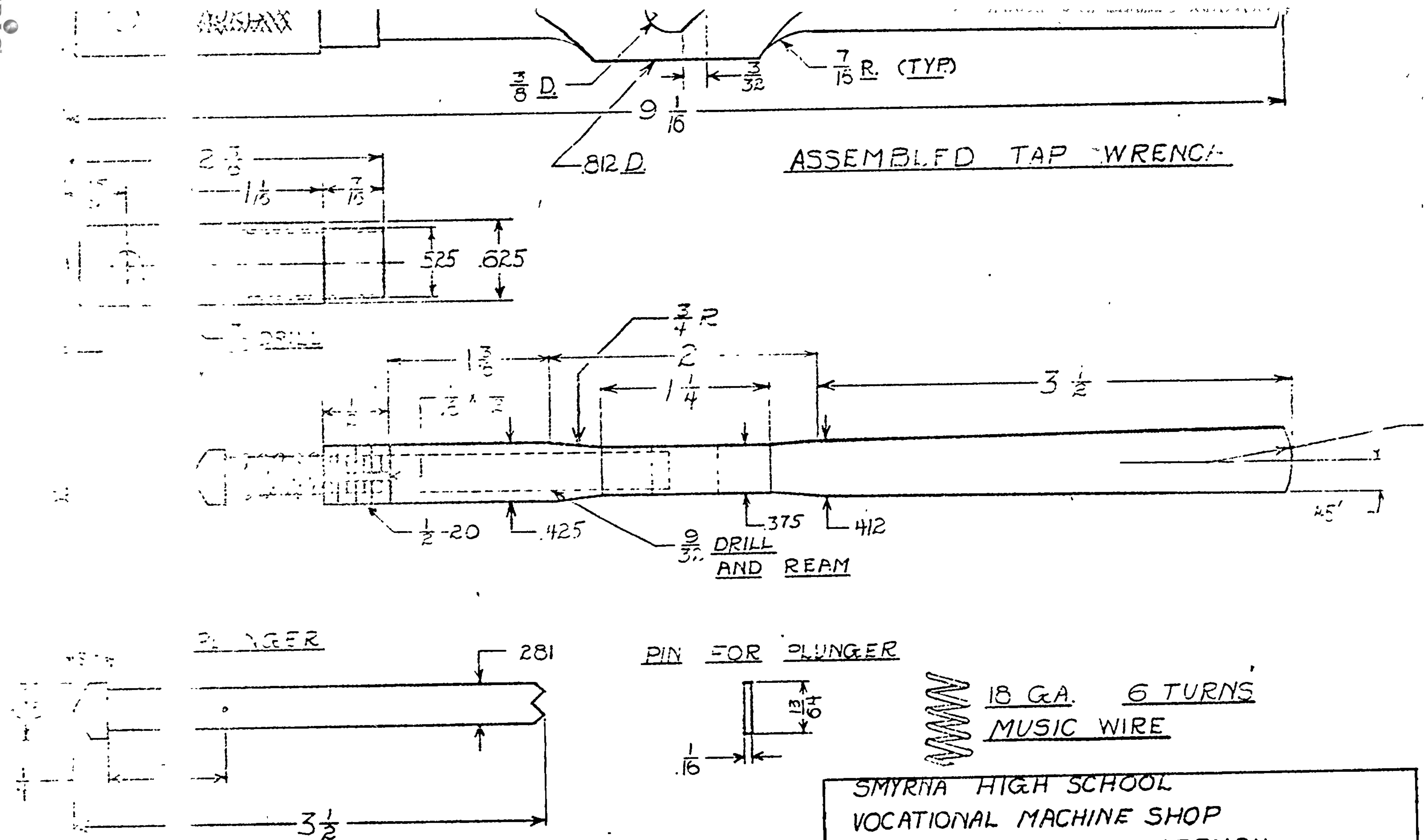
7. Arrange lathe to cut 6 threads per inch, left hand.

   Note: The wire should be stretched at right angles to the rod when winding begins. When spring is wound, back lathe spindle up by hand until tension is relieved, before cutting wire.

8. Wind spring the same as cutting a left hand thread.

9. Cut wire close to hook and remove spring from rod. Cut spring to length and grind ends.

10. Inspect as per drawing.



SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
NAME OF PART TAP WRENCH
PROJECT No. 18 QTY. REQ'D. 1
MAT'L 1018 H.T. S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.002 FRAC. ± 1/64 ANG. ± 1°
SCALE 1=1" DATE 1-14-71 DR. BY

JOB SHEET

TITLE: TO MAKE A DIE STOCK

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in layout and machining of parts and assembly of parts after machining.

INFORMATION: The die stock is used to hold a die for cutting threads of smaller sizes. It holds button dies to cut threads while holding the work in a bench vise. These threads are not as accurate as when they are cut on the lathe.

SPECIFICATIONS:

DIE STOCK

MATERIAL: Cold Rolled Steel
Body 1 3/4" diameter x 7/8" long.
Handle ½" diameter x 5 3/8" long.
Guide 1" diameter x 3/4" long.

TOOLS AND EQUIPMENT: Layout dye, combination square, center-drill, appropriate drills, appropriate taps, and dies, 8" mill smooth file, scribe, ball peen hammer, center punch, ½" counterbore, steel rule, hacksaw, micrometer and hermaphrodite calipers.

PROCEDURE:

BODY

1. Select stock as specified.
2. Cut to length on power hacksaw.
3. Face end square.
4. Center drill.
5. Drill [illegible] diameter hole 1" deep.
6. [illegible] " deep.

[illegible]

8. Reverse stock in lathe.

9. Face end square and to length.

10. Bore .814" diameter x ¼" deep.

    Note: Check inside corner for squarness.

11. Mount work on 3/4" diameter lathe mandral.

12. Turn outside diameter to 1 5/8".

13. Medium knurl as per drawing.

14. Break sharp edges to 1/32" radius.

15. Layout all holes as per drawing.

16. Drill two holes with #20 drill.

17. Tap with 10-32-NF.

18. Drill two holes with 17/64" drill.

19. Tap with 5/16-24-NF.

20. Counterbore ½" diameter x 1/32" deep.

21. Inspect as per drawing.

HANDLE 2 REQ

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Face end square.

4. Centerdrill end and place center in tailstock in part.

5. True part.

6. Knurl end of handle to length of 1 7/8".

7. Reverse stock in lathe.

   Note: Protect knurl with brass.

8. Face end square and centerdrill.

9. [illegible] as per drawing.

10. Turn taper.

11. Cut 5/16-24-NF threads.

12. Machine center out of knurled end.

13. File 1/16" crown on knurled end.

14. Layout 1/8" diameter hole at knurled end.

15. Drill 1/8" diameter hole.

16. Break all sharp edges and polish.

17. Inspect as per drawing.

GUIDE 6 REQ'D

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Face end square.

4. Drill hole "A" as per chart x 3/4" deep.

5. Turn 1.000" diameter.

6. Turn .812" diameter.

7. Cut off to 5/8" long.

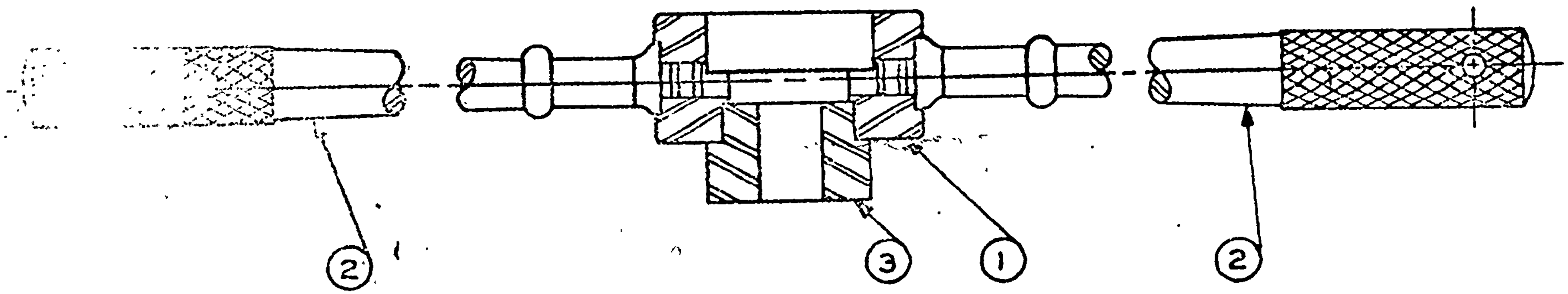
8. Face end square.

9. Break all sharp edges.

10. File or machine 1/4" flat on side.

11. Polish all over.

12. Inspect as per drawing.



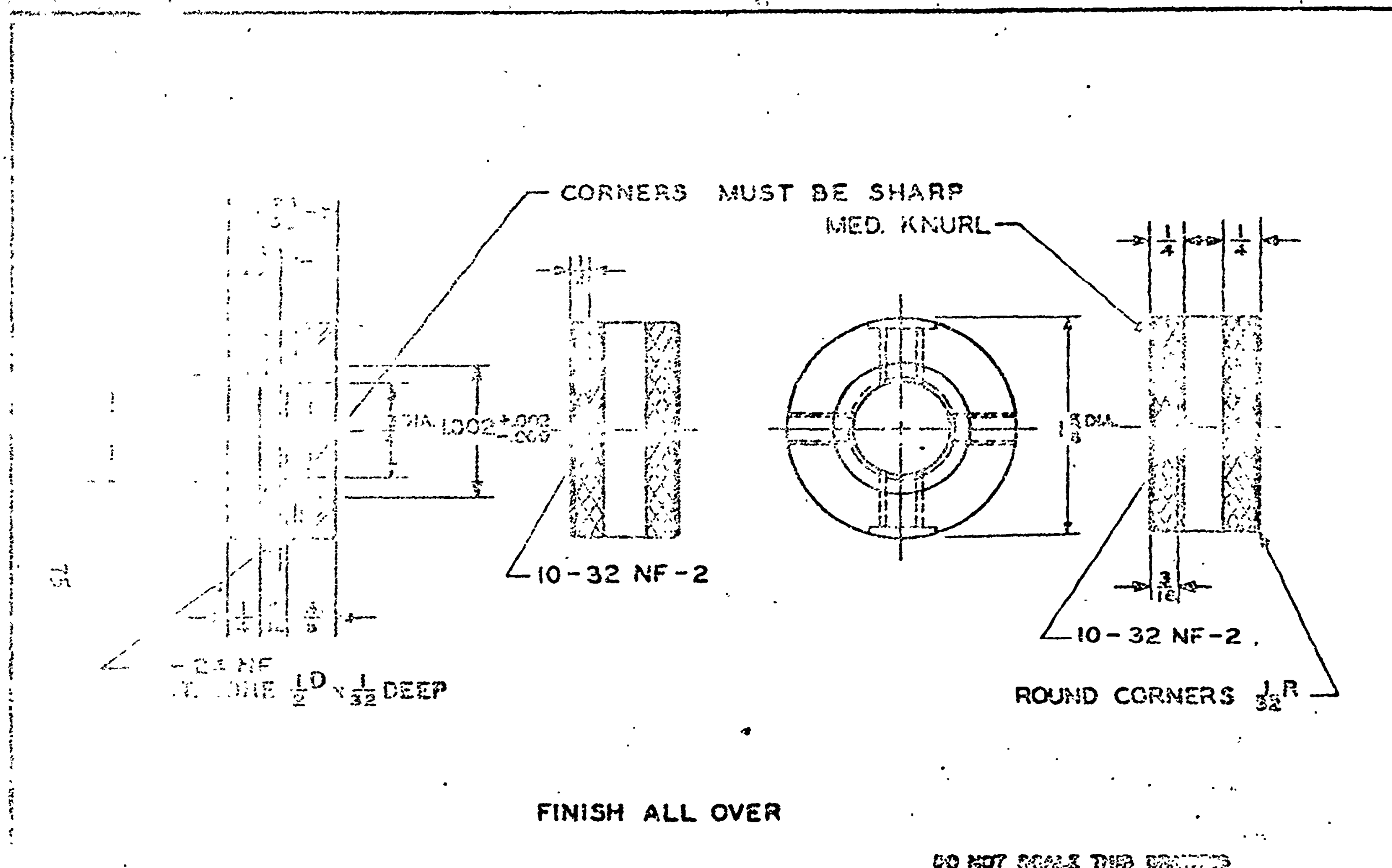
74

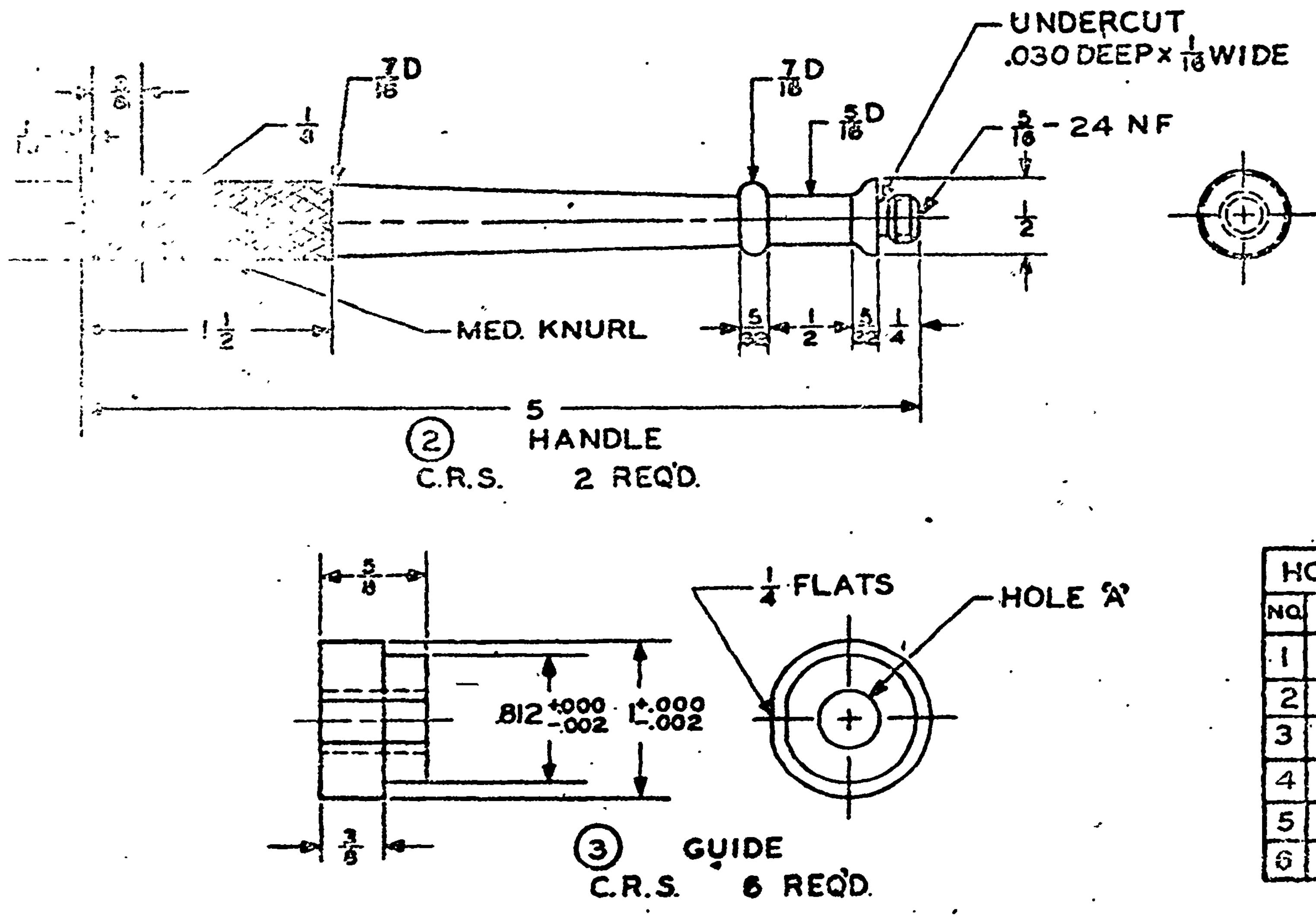
ITEM	QTY	DWG. NO	PART NAME	MATL	SIZE
3	1	A 3	GUIDE	C.R.S	$1 D \times \frac{5}{8}$
2	2	A 3	HANDLE	C.R.S	$\frac{1}{2} D \times 5$
1	1	A 2	BODY	C.R.S	$1\frac{5}{8} D \times \frac{25}{32}$

SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART DIE STOCK
PROJECT No. 10 QTY. REQ'D 1
MAT'L C.R.S. H.T. C.H S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.003 FRAC. ±1/64 ANG. ±1/2
SCALE FULL SHEET 1 OF 3
DR. BY H G DATE
NOTE
CHANGE

DO NOT SCALE THIS DRAWING


CORNERS MUST BE SHARP
MED. KNURL
DIA. 1.002
10-32 NF-2
10-32 NF-2
ROUND CORNERS $\frac{1}{32}$R
$\frac{1}{2}$D x $\frac{1}{32}$ DEEP
FINISH ALL OVER
DO NOT SCALE THIS DRAWING



HOLE 'A'	
NO.	SIZE
1	1/8
2	3/16
3	1/4
4	5/16
5	3/8
6	7/16

FINISH ALL OVER

DO NOT SCALE THIS DRAWING

JOB SHEET

BEST COPY AVAILABLE

TITLE: MACHINIST'S JACK SCREW

UNIT: LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in straight turning, shoulder cutting, recess cutting and internal and external threading.

INFORMATION: A machinist's jack is used in order to have work level. It is sometimes necessary to use a jack to support at some point of the work.

SPECIFICATIONS:

MACHINIST'S JACK SCREW

MATERIAL: Base Cold Rolled Steel 1" Hex x 1 1/8" long.
Sleeve Cold Rolled Steel 7/8" Hex x 1¼" long.
Screw Cold Rolled Steel 5/8" Hex x 2 11/16" long.
Swivel Cole Rolled Steel 13/16" diameter x 2" long.

TOOLS AND EQUIPMENT: Power hacksaw, layout dye, facing tool, tool holder, 3-jaw chuck, center drill, drill chuck, boreing bar, 3/4-16-NF tap, 23/64" drill 7/16-14-NC tap, radius form tool, form tool, 6/32" drill and hand hacksaw.

PROCEDURE:

BASE

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Mount work in 3-jaw chuck.
4. Face end of stock.
5. Reverse stock in chuck.
6. Face end to length as per drawing.
7. Center drill end of stock.
8. [illegible] drill.

[illegible]

10. Set lathe for proper speed and feed for threading.

11. Set compound rest for internal threading.

12. Cut internal thread.

    Note: Allow .005"-.010" stock for tap to remove.

    Note: Use plenty of cutting oil.

13. Finish thread with 3/4-16-NF tap.

    Note: Tap on lathe, with center mounted in tailstock inserted into center of tap.

14. Chamfer corners of hex.

15. Chamfer ends of threads.

16. Inspect as per drawing.

PROCEDURE:

SLEEVE

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck.

4. Face end of stock.

5. Rough turn diameter to be threaded.

6. Face sholder and cut threading recess as per drawing.

7. Centerdrill end of stock.

8. Drill with 23/64 drill as per drawing.

9. Start thread with tap as per drawing to a depth of 1/2".

10. Chamfer end of thread.

11. [illegible] portion to be threaded.

    Note: [illegible] .0[illegible]" undersize to insure

13. Chamfer outside corners of hex section and end of thread.

14. Place work in bench vise and finish tapping by hand.

    Note: Use plenty of cutting oil.

15. Screw sleeve into base with shoulder of sleeve against top of base. Place work in lathe chuck gripping on base.

16. Face end and rough face shoulder.

17. Finish face shoulder, using tool ground to form fillet. File corner.

18. Chamfer outside corners of hex.

19. Chamfer end of threads.

20. Inspect as per drawing.

PROCEDURE:

SCREW

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck.

4. Face end of stock.

5. Reverse stock in chuck.

6. Face end of stock to length.

7. Rough turn diameter to be threaded.

8. Face shoulder and cut threading recess as per drawing

9. Finish turn section to be threaded.

    Note: Turn diameter .005" undersize to insure free fit.

10. [illegible] for proper speed and feed to cut [illegible]

    [illegible] oil.

11. Screw the finished thread into sleeve with shoulder of screw againist top of sleeve. Screw sleeve into Base and place in 3-jaw chuck.

12. Rough turn diameter of ball.

13. Turn fillet next to hex section using a forming tool.

14. Face shoulder and form fillet in one operation, using forming tool.

15. Finish turn ball, using forming tools.

16. Inspect as per drawing.

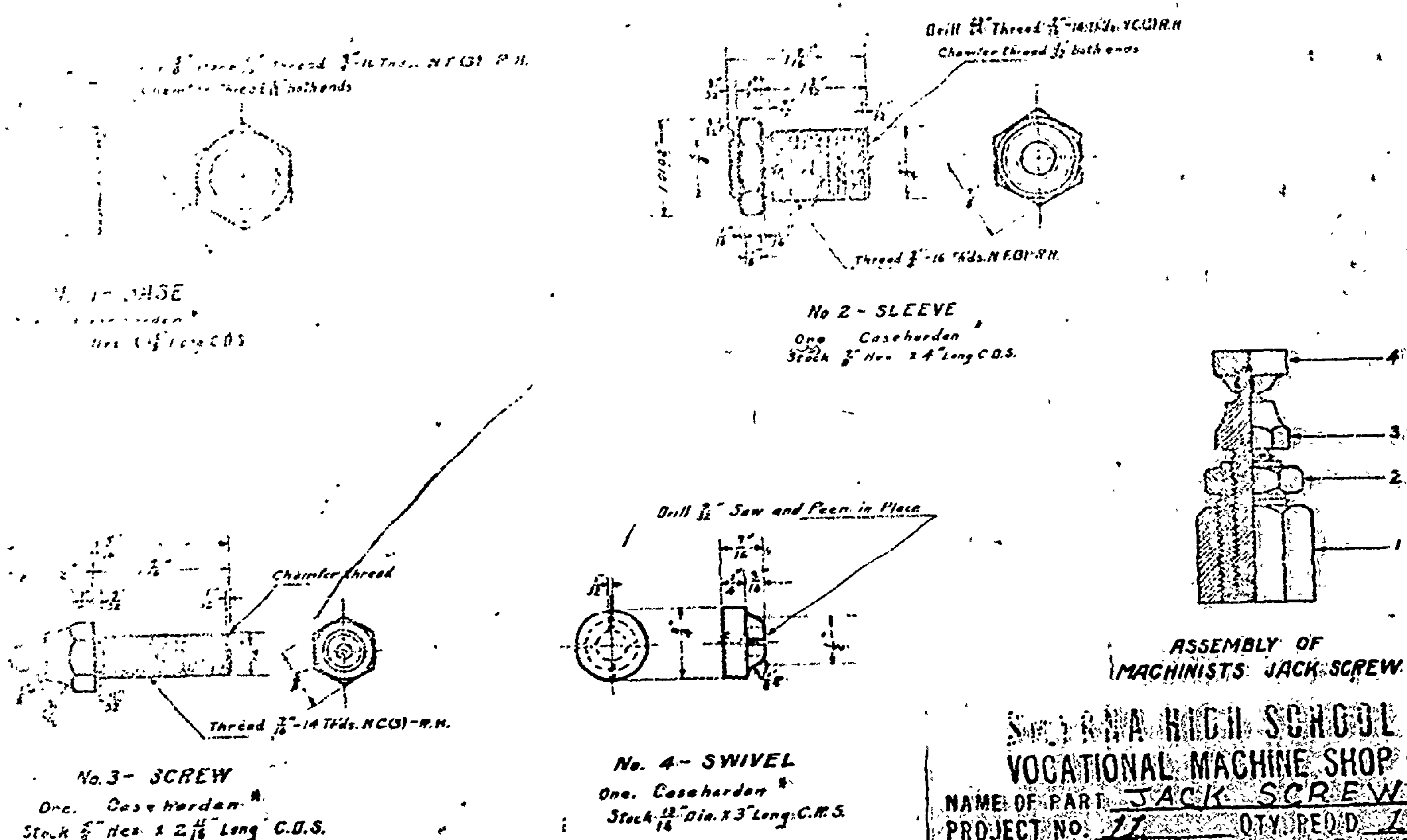
PROCEDURE:

SWIVEL

1. Select stock as specified.
2. Cut to length with power hacksaw.
3. Mount in 3-jaw chuck.
4. Face end.
5. Rough turn diameter.
6. Center drill and drill 9/32". 3/16" deep.
7. Finish turn diameter.
8. Face shoulder and form fillet.
9. Radius end as per drawing with 8" mill smooth file.
10. Cut off to length as per drawing, allow 1/32 stock to face end.
11. Mount in 3-jaw chuck.
12. Face end to correct length.
13. [illegible] slots with hand hacksaw as per drawing.

[illegible] with soft-jaws.

[illegible] and-peen in place.



SONOMA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART JACK SCREW
PROJECT NO. 11 QTY. REQ'D 1
MAT'L C.R.S. H.T. C.H. S.F.
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.003 FRAC. ±1/64 ANG. ±1/2
SCALE 1/2 SHEET 1 OF 1
DR. BY C.D. DATE
NOTE
CHANGE

JOB SHEET

TITLE: TO MAKE A SURFACE GAUGE

UNIT: LATHE

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in layout and all phases of machine operations and assembly of parts.

INFORMATION: A surface gauge is used for layout purpose with a surface plate.

SPECIFICATIONS:

SURFACE GAGE

MATERIAL:

Part No. 1	3½" diameter x 3" long C. R. S.
Part No. 2	1¼" diameter x 2" long C. R. S.
Part No. 3	5/16" Diameter x 10½" long Drill Rod
Part No. 4	5/8" diameter x 2½" long C. R. S.
Part No. 5	3/16" diameter x 1" long. C. R. S.
Part No. 6	7/8" diameter x 2" long. C. R. S.
Part No. 7	¼" diameter x 1½" long. C. R. S.
Part No. 8	7/16" diameter x 1½" long C. R. S.
Part No. 9	7/16" x 7/16" x 1" C. R. S.
Part No. 10	1/8" diameter x 5¼" long Drill Rod

TOOLS AND EQUIPMENT

TO BE DETERMINED BY STUDENT

PROCEDURES:

BASE

1. [illegible]
2. [illegible]

3. Face end square.
4. Drill and Ream all holes
5. Machine as per drawing.
6. Cut to length.
7. Cut recess in base.
8. Drill hole for spindle key
9. Drill and tap 10-24-NC hole.
10. Inspect as per drawing.

ADJUSTING COLLAR

1. Slect stock as specified.
2. Cut to length on power hacksaw.
3. Face end and center drill.
4. Rough turn all diameters.
5. Finish turn all diameters.
6. Knurl large diameter.
7. Drill hole to be taped and start tap in lathe.

   NOTE: Use plenty of oil.
8. Chamfer end of thread.
9. Face shoulder to length.
10. Check for fit of 3/4" diameter in to reamed hole of base.
11. Mark location of recess on small diamter.
12. Cut recess with cut off tool.
13. [illegible]
14. [illegible]

SPINDLE

1. Select stock as specified.
2. Cut to length on power hacksaw.
3. Mount work in collect and face end square and to length.
4. Machine 5/16-24-NF threads and chamfer.
5. Polish all over.
6. Mill keyseat as per drawing.
7. Inspect as per drawing.

THUMB SCREW

1. Select stock as specified.
2. Cut to length on power hacksaw.
3. Face end square and center drill.
4. Use lathe center in tail stock.
5. Rough turn large diameter.
6. Rough turn small diameter.
7. Finish turn large daimeter.
8. Knurl large diameter.
9. Use form tool and machine radius.
10. Finish turn small diameter and face shoulder to length.
11. Machine threads on lathe.
12. Turn pilot section on end of threaded end, allowing ½" for removing center hole.
13. Saw off (with a hacksaw) approximately 7/32" of pilot section.
14. Face [illegible] length.
15. [illegible] th.
16. [illegible]

SPINDLE KEY

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. File angles and flats on stock as per drawing.

4. Place key in base and check for sliding fit in keyway of spindle.

5. File and polish top of key flush with base.

6. Inspect as per drawing.

CLAMP NUT

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Face end square.

4. Center drill, drill and tap 10-24-NC hole.

5. Rough turn large diameter

6. Rough turn small diameter.

7. Finish turn large diameter.

8. Knurl large diameter.

9. Finish turn small diameter using form tool.

10. Face shoulder.

11. Cut to length.

NOTE:. Leave clamp nut unfinished for the present and make clamp screw and use it as a stub arbor to face end of clamp nut.

CLAMP SCREW

1. Select stock as specified.

2. Cut to length on power hacksaw.

4. Turn diameter as per drawing.

5. Machine threads as per drawing.

6. Chamfer thread.

7. Cut to length and face end square.

8. Place clamp screw in collect with threads projecting 3/8", screw clamp nut on threads and face end square.

   NOTE: Do not drill #30 hole.

9. Inspect as per drawing.

NEEDLE SLEEVE

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Face end square, centerdrill and drill with 3/16" drill ½" deep.

4. Turn diameter to 3/8".

5. Cut to length and face end square.

6. Layout and center punch hole to be drilled through diameter.

7. Place needle sleeve and clamp screw together in correct position.

8. Transfer to drill press and drill holes as per drawing.

9. Remove all burrs.

10. Inspect as per drawing.

CLAMP

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Square ends and machine to length.

4. Machine or file radius.

5. Drill [illegible]

6. Ream 19/64" hole to 5/16"

7. Place in mill vise and saw 1/32" slot.

8. Inspect as per drawing.

NEEDLE

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Grind ends to point as per drawing.

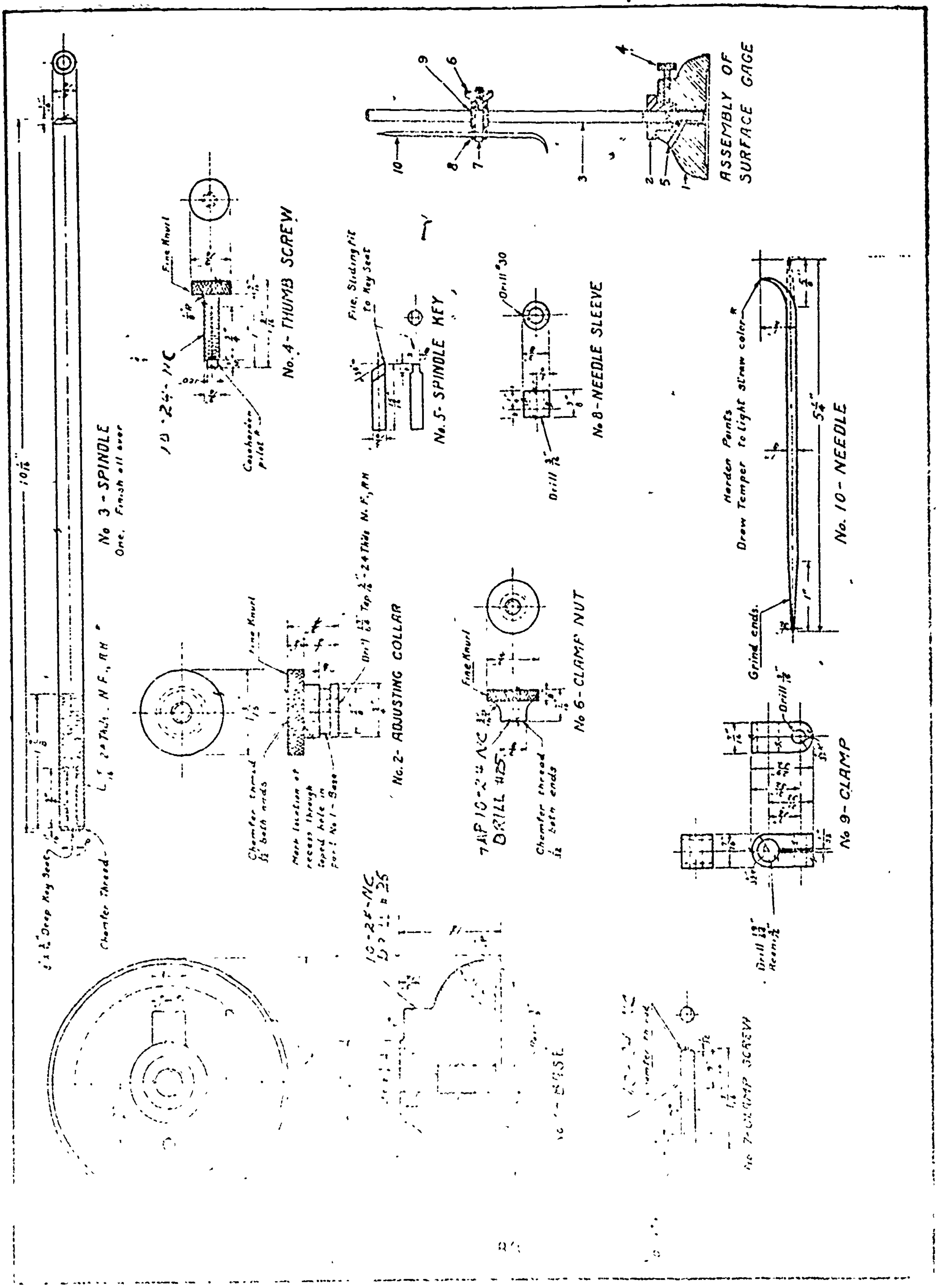
4. Bend hook end as per drawing.

5. Harden and draw temper.

6. Polish all over.

7. Inspect as per drawing.

BEST COPY AVAILABLE



JOB SHEET

BEST COPY AVAILABLE

TITLE: TO MAKE A MACHINIST VISE

UNIT: MILLING MACHINE AND LATHE

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in layout and all phases of machine operations and assembly of parts to a close tolerance.

INFORMATION: The machinist vise is a very useful tool in holding small parts to be machined or ground on the surface grinder and for bench work.

SPECIFICATIONS:

MACHINISTS VISE

MATERIAL: Cold Rolled Steel

Body - 1 3/4" x $1\frac{1}{2}$" x $4\frac{1}{4}$" long.

Jaw - 1 3/4" x 1 3/16" x 7/8"

Screw - 7/8" diameter x $5\frac{1}{4}$" long.

Guide plate - 1/8" x $\frac{1}{2}$" x 1" long.

Hand wheel - 2" diameter x $2\frac{1}{4}$" long.

TOOLS AND EQUIPMENT: Layout dye, vernier height gage, approrpiate milling cutters, drills and taps, milling machine, scribe, hermaphrodite calipers, combination square, engine lathe, facing tool, right hand turning tool, knurling tool, threading tool, center gage, index head, and shaper.

PROCEDURE:

BODY

1. Select stock as specified.
2. Cut to length on power hacksaw.
3. Square work in shaper and machine to dimensions [illegible]

5. Machine opening .812" deep x 1 3/16" long.

6. Machine recesses with 1/16" ball end mill.

7. Machine ½" and ¼" grooves in bottom of vise as per drawing.

8. Place work in vise on mill with hole to be drilled up.

   Note: Check for squarness with square.

9. Find center of hole to be drilled with a wiggler.

10. Centerdrill and drill hole and tap 3/8-16-NC.

    Note: Start tap in machine to insure squarness.

11. Machine angle ¼" x 45 degree on back of vise.

12. Machine angle on solid jaw of vise.

13. Break all sharp edges.

14. Inspect as per drawing.

15. Case harden and draw temper.

16. Grind all over.

JAW

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Square work in shaper and machine to dimensions as per drawing.

4. Layout as per drawing.

5. Center drill, drill and tap 8-32-NC holes in bottom of jaw.

6. Drill 5/16" diameter hole in back of jaw.

7. Machine ½" radius as per drawing.

8. Machine [illegible] x ½" guide on bottom of jaw.

   [illegible]

9. Break all sharp edges.

10. Inspect as per drawing.

11. Caseharden and draw temper.

12. Grind all over to fit body.

SCREW

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Face both end square and center drill.

   Note: Leave 3/8" on each end to remove centerdrill.

4. Turn threaded diameter to .370" x 4 1/8" long.

5. Turn large diameter to ~~.750"~~

6. Turn small diameter to .480".

7. Medium knurl large diameter.

8. Chase threads on lathe 3/8-16-NC.

   Note: Use plenty of oil.

9. Machine end of threads to .312" to fit movable jaw.

10. Remove center drill and machine 59 degree angle on end of screw.

11. Remove center drill from .480" diameter end.

12. Mount work in index head and machine .375" square on end of screw.

13. Break all sharp edges and champher threads.

14. Inspect as per drawing.

15. Caseharden and draw temper.

16. Polish all over.

[illegible]

1. [illegible]

3. Square ends and machine as per drawing.

4. Layout holes as per drawing.

5. Drill and counter sink holes.

6. Break all sharp edges.

7. Grind two sides.

8. Inspect as per drawing.

HAND WHEEL

1. Select stock as specified.

2. Cut to length on power hacksaw.

3. Face end square.

4. Turn small diameter to 1.000 x .375".

5. Turn large diameter to 1.875" x 5/8".

6. Medium knurl on large diameter.

7. Center drill and drill 1" deep.

8. Cut to length.

9. Face end square.

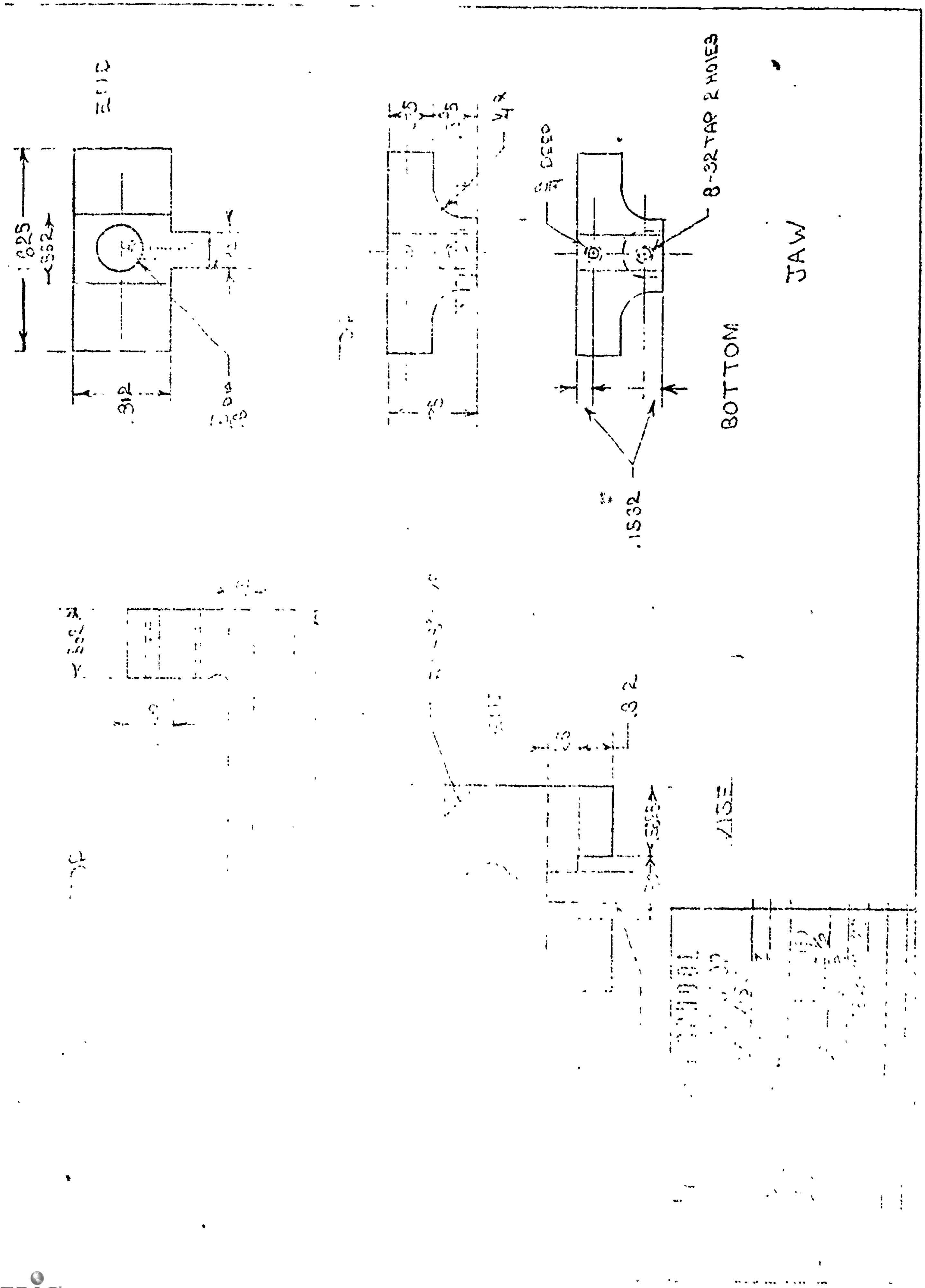
10. Machine 20 degree angle on inside of hand wheel.

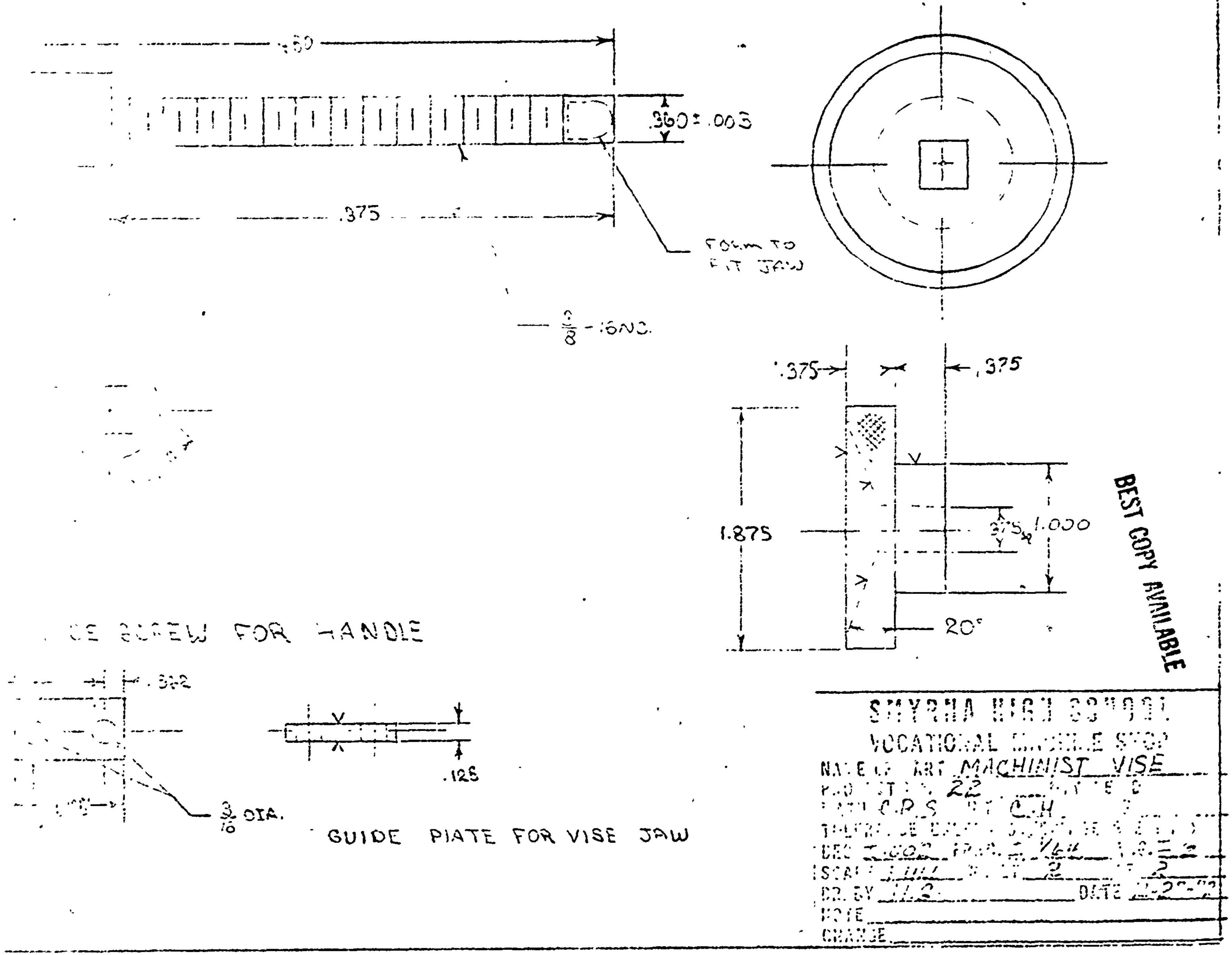
11. Broach 3/8" square hole on hand arbor press.

    Note: Use oil and check for squarness.

12. Polish all over.

13. Inspect as per drawing.


JAW
BOTTOM
8-32 TAP 2 HOLES
1.625
.312
.1532
.32


.360 ±.003
.375
FORM TO FIT JAW
3/8 - 16NC
.375
.375
1.875
.375
1.000
20°
BEST COPY AVAILABLE
SCREW FOR HANDLE
.125
3/16 DIA.
GUIDE PIATE FOR VISE JAW
SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
NAME OF ART MACHINIST VISE
22
C.R.S.
C.H.
DATE
NOTE
CHANGE

JOB SHEET

TITLE: TO MAKE A 5" SINE BAR

UNIT: LATHE, MILLING MACHINE AND GRINDER

OCCUPATION: MACHINIST

OBJECTIVE: To develope skills and sequence on machining operations when several machines are used and precision is of the main importance.

INFORMATION: The sine bar is an instrument of precision used by the toolmaker in laying out, setting, testing and otherwise dealing with angular work which requires a close degree of accuracy in its dimensions. It consists of a bar of steel with two rollers or logs of equal diameter secured near the ends of the bar and having their center on line exactly parallel with the edges of the sine bar.

SPECIFICATIONS:

5" SINE BAR

MATERIAL: Cold Rolled Steel 1"x1 1/8" x 6 3/8" long
Cold Rolled Steel 3/4" diameter 3¼" long.
Cold Rolled Steel ½" x 1" x 1 1/8" long.

TOOLS AND EQUIPMENT: Power hacksaw, layout fluid, combination square, bevel protractor, 8" mill smooth file, center punch, ball peen hammer, scribe, 3/16" drill, [illegible] tap, 7/32 counter bore, 5/16", [illegible] 7/16" drill, No. 36 drill, steel rule, [illegible], [illegible] tap, No. 26 drill, vernier height gage and angle plate.

PROCEDURE:

BODY

1. Select stock as specified.

2. [illegible] with power hacksaw.

3. [illegible] side, top and one end square.

4. [illegible]

6. Layout six relief holed.

7. Machine to layout lines.

   Note: Leave .025" for grind all over.

8. Drill holes as per drawing.

9. Counter sink both sides.

10. Drill and tap two holes for 8-32-NC. screws.

11. Complete layout.

12. Drill two 1/8" diameter relief holes.

13. Machine to layout lines.

    Note: Leave grind stock.

14. Set up stock in mill and drill. 166 and counter bore for 8-32-NC screws.

15. Harden and temper.

16. Grind all over as per drawing.

17. Break all sharp edges.

18. Inspect as per drawing.

ROLLER

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Mount work in 3-jaw chuck or collect on lathe.

4. Face ends square and centerdrill.

5. Mount work between centers.

6. Layout as per drawing.

7. Rough turn to dimensions before heat treat.

8. [illegible] cut with 1/8" cutoff tool.

9. [illegible] to be drilled as per drawing.

[illegible] two holes 8-3[illegible]

11. Harden and temper.

12. Clean scale and dirt out of center holes.

13. Place work between centers on clyndrical grinder.

14. Clean up on grinder.

    Note: Check for taper.

15. Grind to finished dimensions.

16. Mount work in V-Block on surface grinder.

17. Cut into and ends off with cut off wheel.

18. Surface grind ends as per drawing.

    Note: Hold work in V-block for squarness.

19. Break all sharp edges.

20. Inspect as per drawing.

END PLATE

1. Select stock as specified.

2. Cut to length with power hacksaw.

3. Square end and side in mill.

4. Layout as per drawing.

5. Center drill and drill two holes .166 diameter.

6. Machine to dimensions.

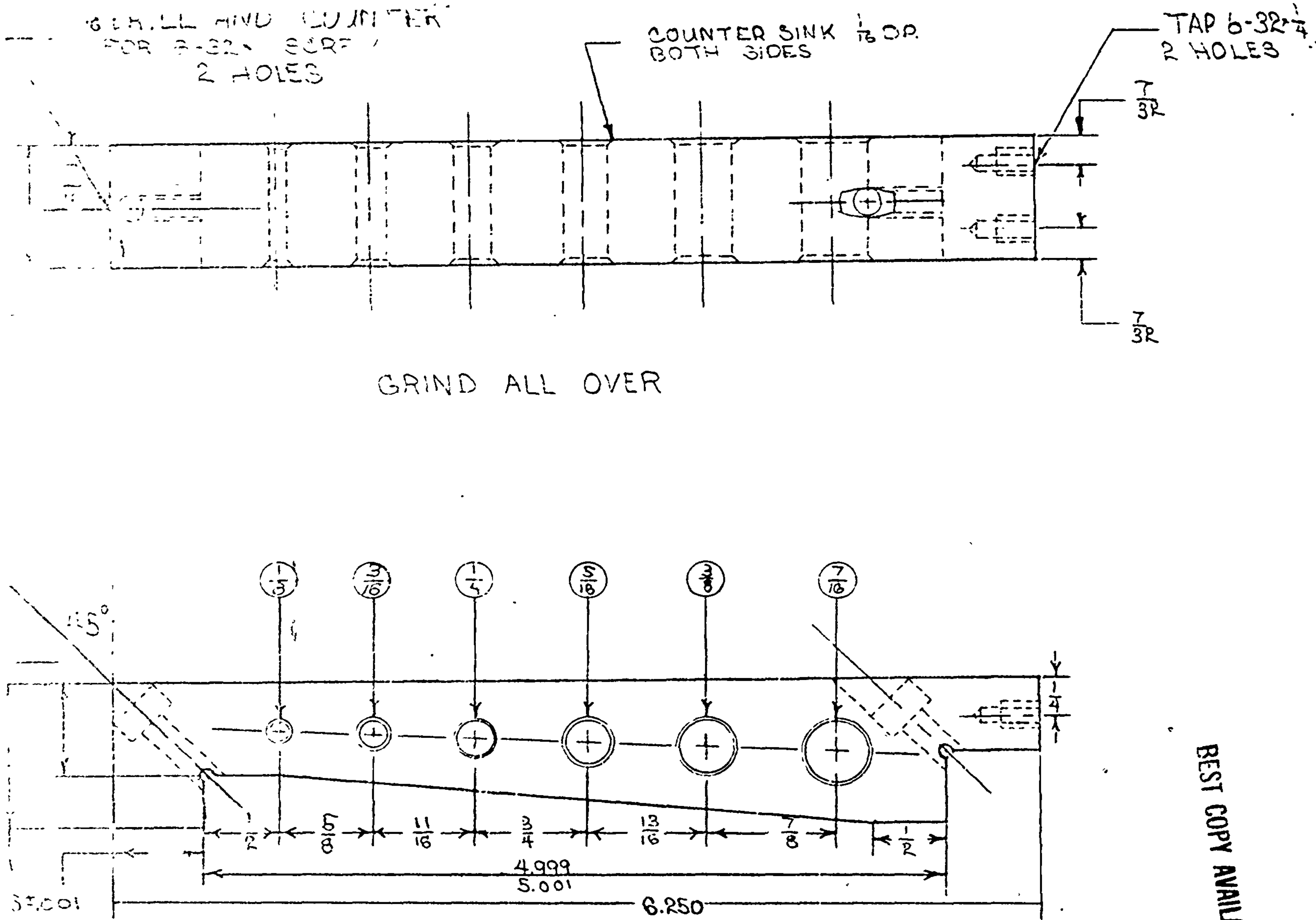
    Note: Leave grind stock.

7. Harden and temper.

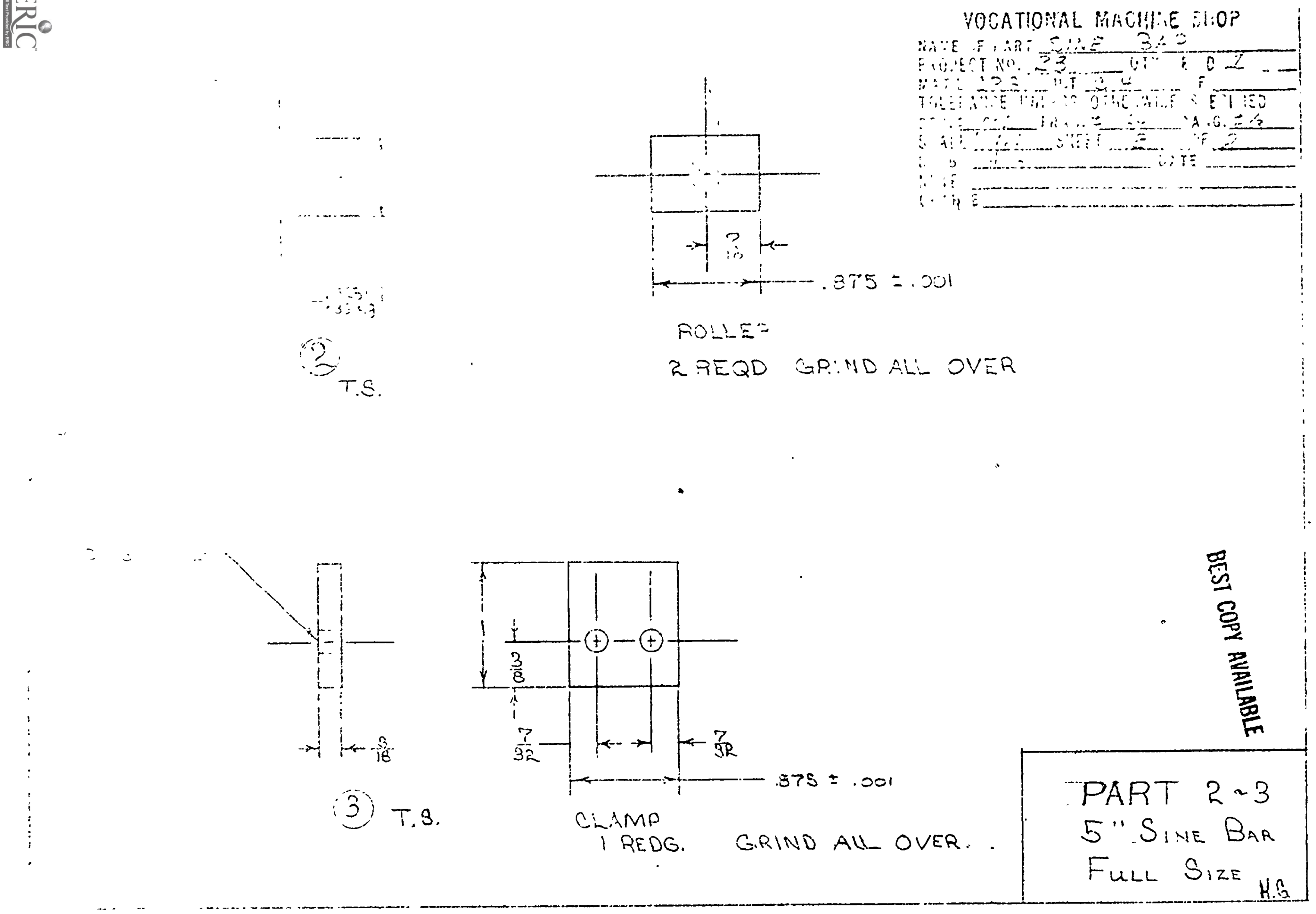
8. [illegible] all over

[illegible] all sharp edges.

[illegible] as per drawing.



BODY 5" SINE BAR SCALE-FULL SIZE



BEST COPY AVAILABLE

JOB SHEET

TITLE: TO MAKE A HAND VISE

UNIT: MILLING MACHINE AND LATHE WORK

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in layout and all phases of machine operations and assembly of parts to a close tolerance.

INFORMATION: The hand vise is a very useful tool in holding small parts for hand work at the bench.

SPECIFICATIONS:

HAND VISE

MATERIAL: HANDLE: 1¼" diameter x 5" long aluminium.

SCREW: ½" diameter x 4" long C.R.S.

SCREW SLEEVE 5/8" diameter x 5/8" long C.R. S.

LEADER PIN 3/8" diameter x 2 15/16" long C.R.S.

HANDLE: ½" diameter x 3½" long C. R. S.

BALLS: 3/8" diameter x 1½" long. C. R. S.

CENTER POST: 3/4" x ½" x 3¾" C. R. S.

JAWS: 5/8" x 1½" x 4 7/8" C. R. S.

TOOLS AND EQUIPMENT: Layout dye, Engine lathe, 3-Jaw chuck, Appropriate lathe tools, appropriate drills, reamers and taps, 4-Jaw chuck, Milling machine, Rotary table, appropriate mill cutters.

PROCEDURES:

TO BE DEVELOPED BY SUTDENT

VOCATIONAL MACHINE SHOP

NAME OF ... VISE ASSEMBLY + HANDL

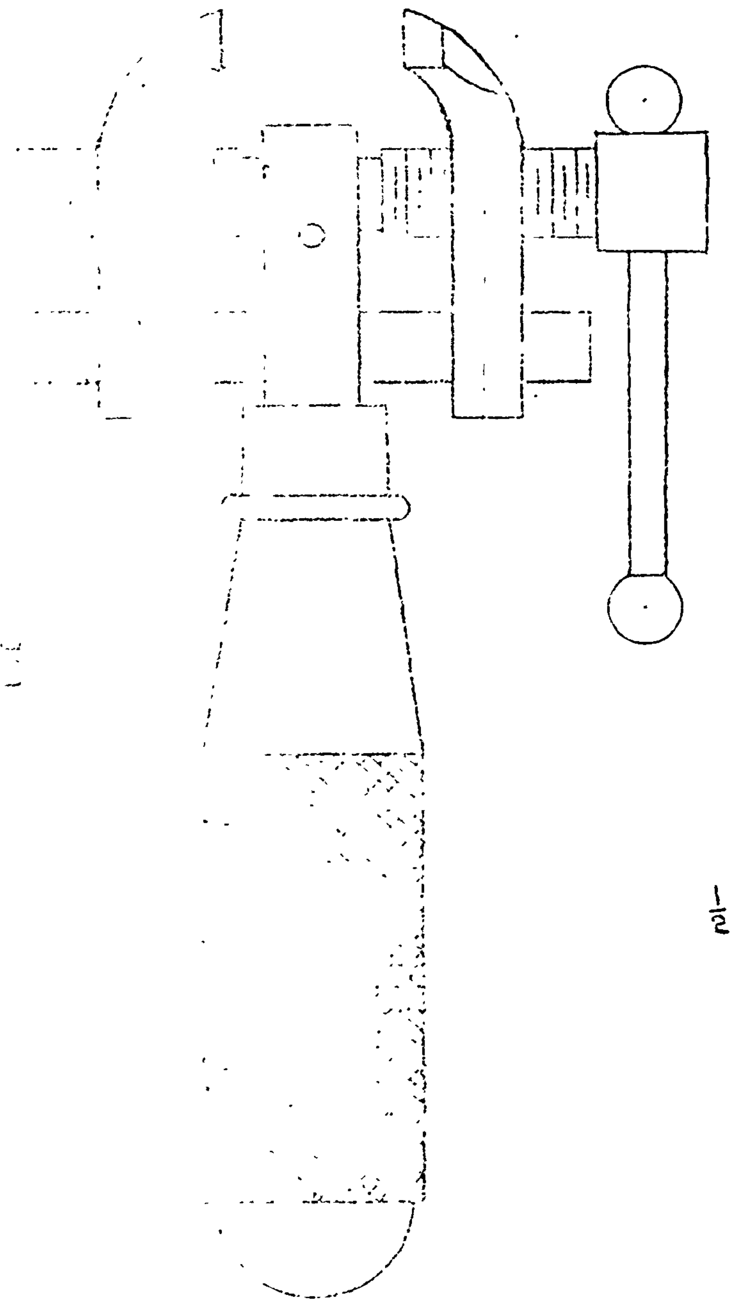
... 1

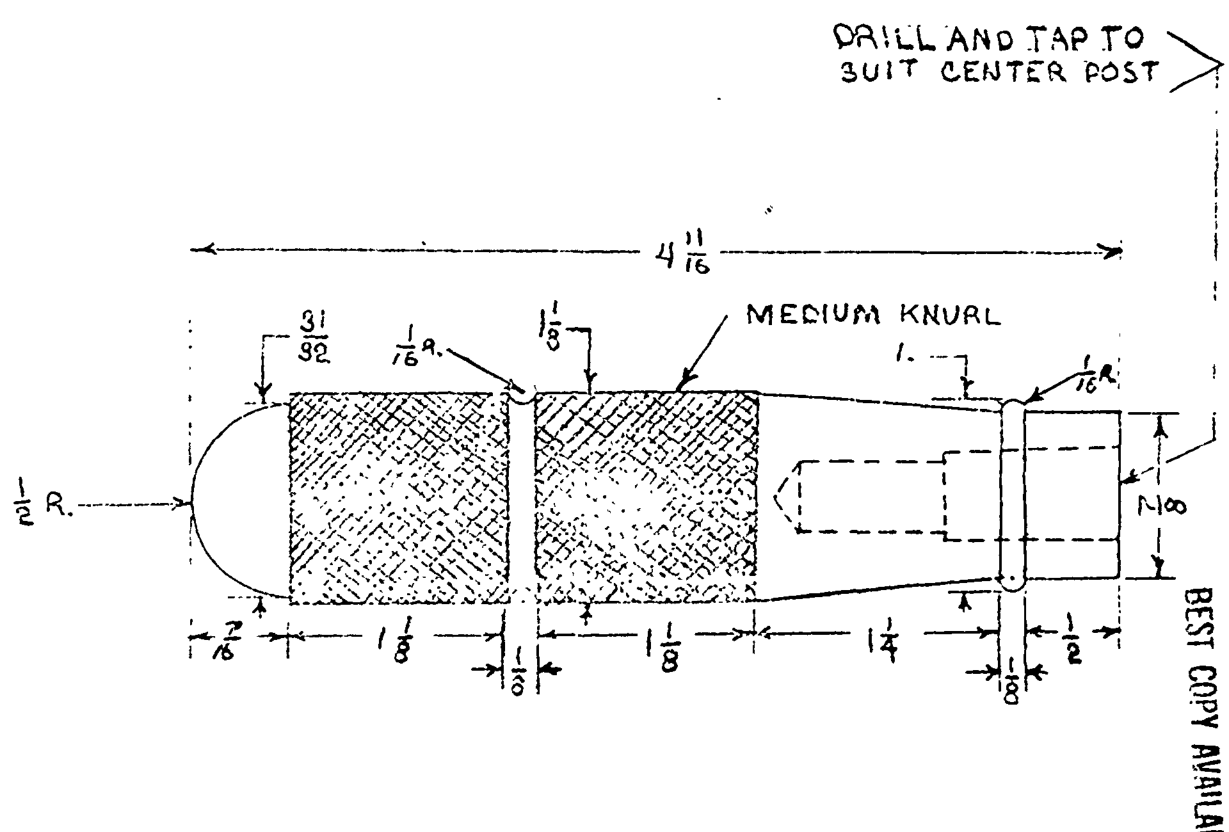
... ALUM

... ±.005 ... ±1/64 ... ±1/2°

SCALE FULL ... 1 ... 4

... H.G ... 1-9-74





HANDLE

BEST COPY AVAILABLE

VOCATIONAL MACHINE SHOP

NAME OF PART JAWS

PROJECT NO. ______ QTY. REQ'D 2

MAT'L. C-1045 H.T. C.H S.F ______

TOLERANCE UNLESS OTHERWISE SPECIFIED

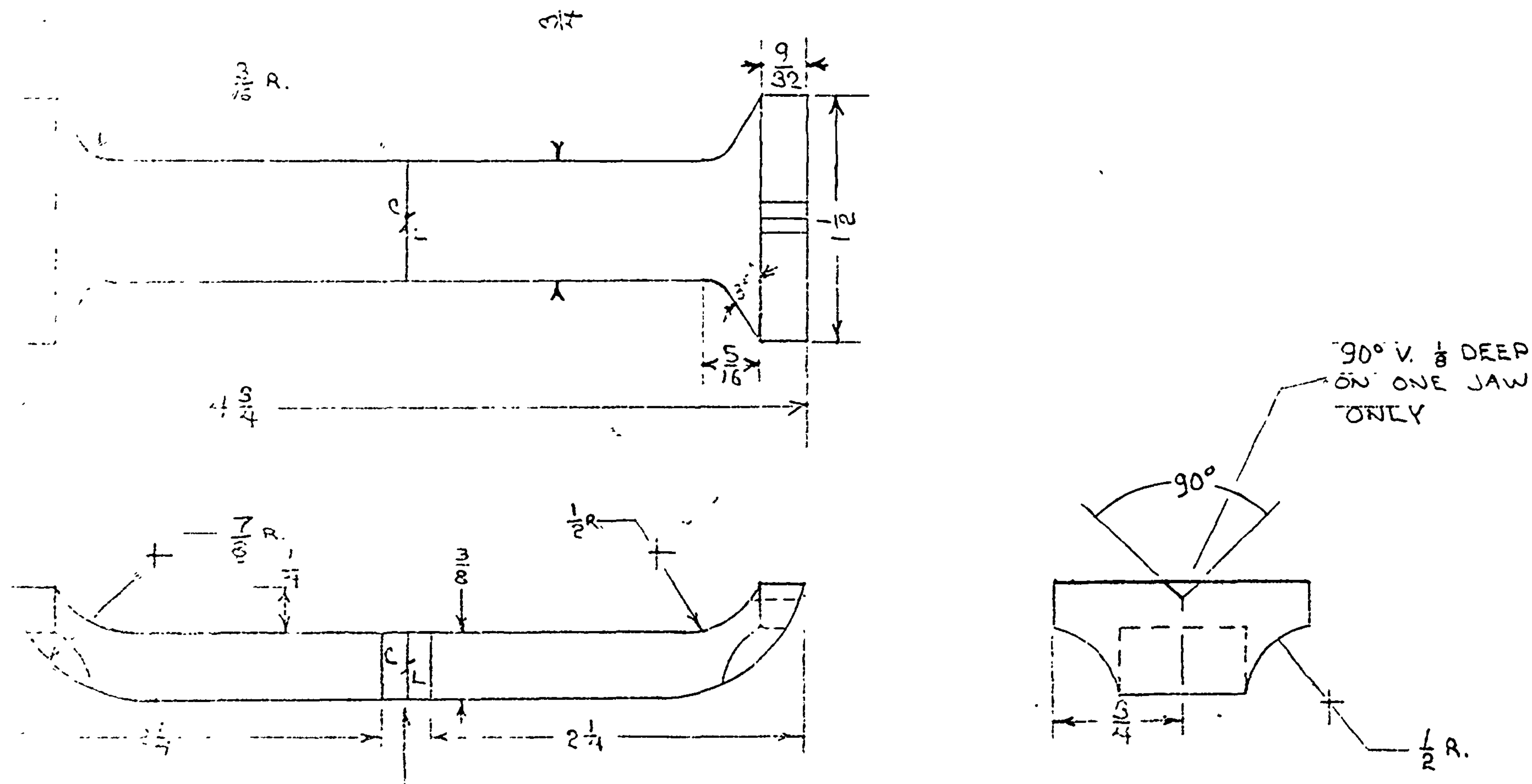
DEC. ±.002 FRAC. ± 1/64 ANG. ±1/2°

SCALE FULL SHEET 2 OF 4

DR BY H.G. DATE ______

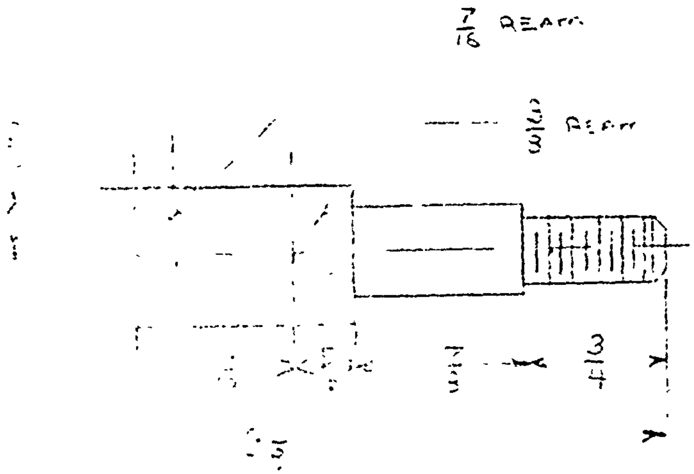
NOTE ______

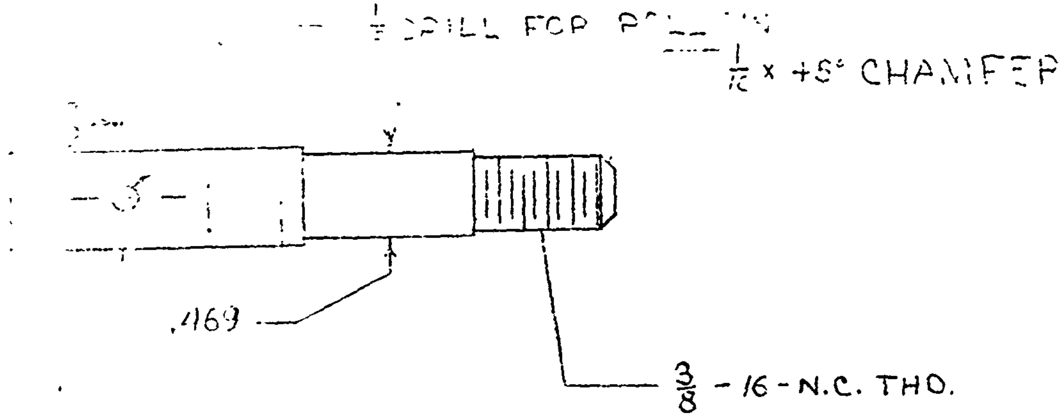
CHANGE ______

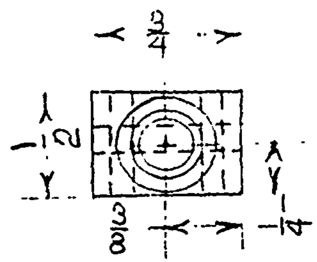


SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

NAME OF PART CENTER POST
PROJECT NO. ______ QTY. RE. D 2
MAT'L. C-1045 H.T. C.H. S.F ______
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ±.002 FRAC. ± 1/64 ANG. ± 1/2°
SCALE FULL SHEET 3 OF 4
DR. BY H G DATE ______
NOTE DRILL HOLE CLAMPED ...
CHANGE ... DWG.

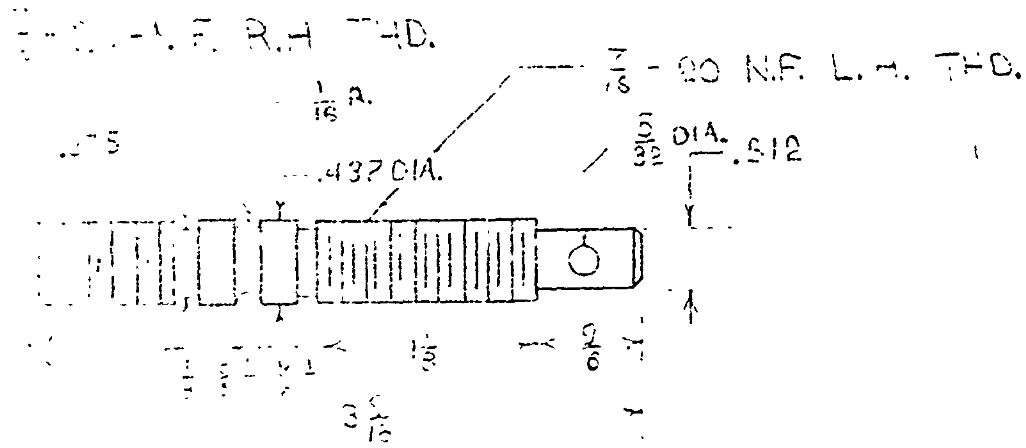




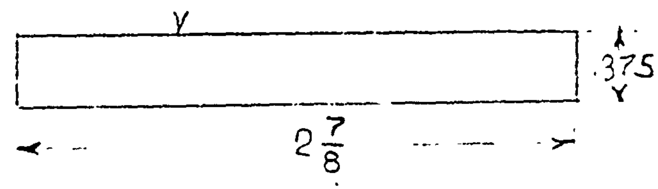


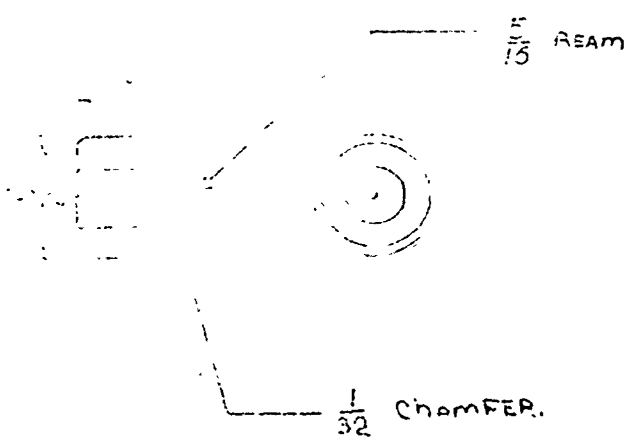
SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP

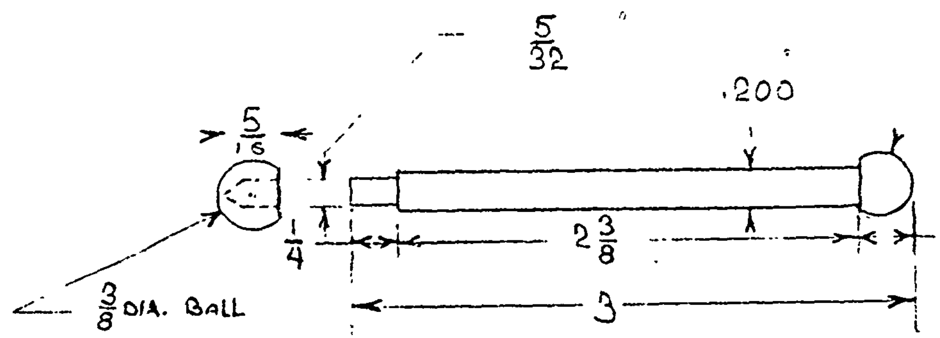
NAME OF PART Screw - Sleeve - Handle
PROJECT No. ________ QTY. REQ'D 1
MAT'L. C 1045 H.T. ________ S.F ________
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC. ± .002 FRAC. ± 1/64 ANG. ± 1/2°
SCALE FULL SHEET ___ OF ___
DR BY H. G. DATE ________
NOTE Drill Sleeve + Screw For ________
CHANGE ________



BRAKE SHARP EDGES







JOB SHEET

TITLE: TO MAKE AN ARBOR PRESS

UNIT:

OCCUPATION: MACHINIST

OBJECTIVE: To develop skills in precision layout and machining operations and machining of rack and pinion gears.

INFORMATION: Arbor presses are mainly used for pressing parts together and for broaching a square or rectangular hole.

REFERENCE:

SPECIFICATIONS:

ARBOR PRESS

MATERIAL: Cold Rolled Steel
Table 2" diameter x 5/8" long
Cover Plate: ¼" x 1" x 2 1/8" long
Table Pin: 5/16" diameter x 1 1/8" long.
Sleeve: 1½" diameter x 2" long
Pinion Gear: 1½" diameter x 2 3/4" long
Gear Shaft Screw: 3/4" diameter x 1½" long
Rack Pad: 7/8" diameter 1½" long
Handle: 3/8" diameter x 5½" long
Handle Ball: 5/8" diameter x 1½" long.
Rack Gear: 5/8" x ½" x 5" long.
Column 1 1/8" x 3 3/8" x 6 1/8"
Base: 3" x 5½" x 1"

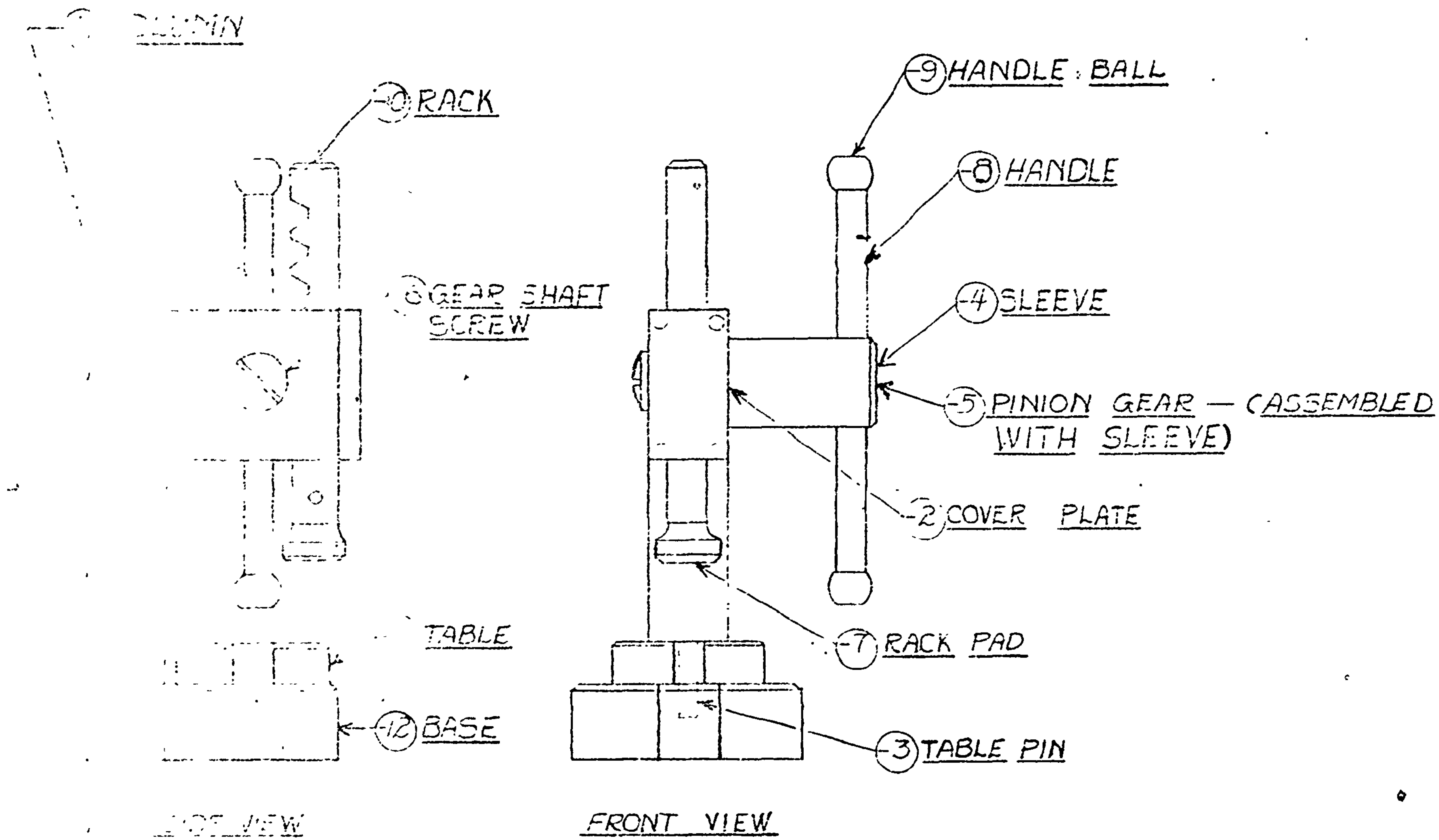
TOOLS AND EQUIPMENT:

TO BE DETERMINED BY STUDENT

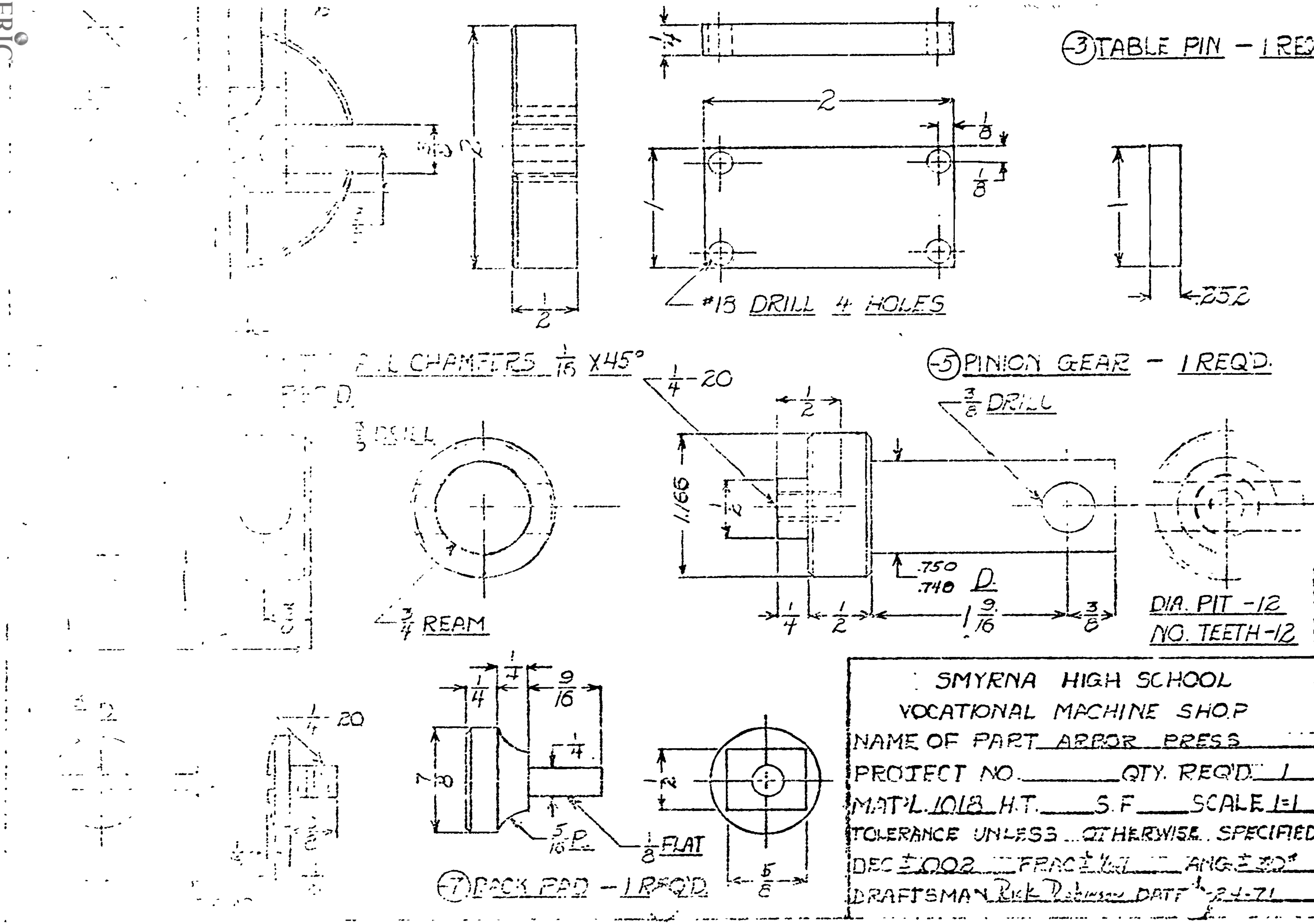
PROCEDURES:

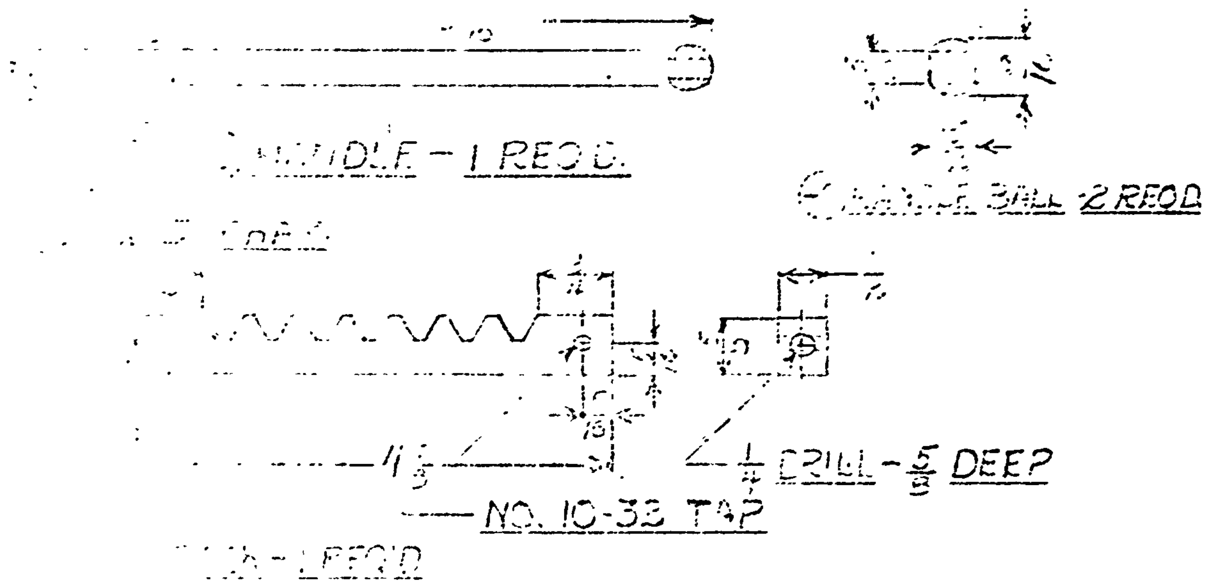
TO BE DEVELOPED BY STUDENT

ASSEMBLED ARBOR PRESS



SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
DRAFTSMAN Rick Robins DATE- 2-25-71


-3 TABLE PIN - 1 REQ'D
2
1/8
1/8
1
#13 DRILL 4 HOLES
1/2
.252
2
1/2
ALL CHAMFERS 1/16 X45°
-5 PINION GEAR - 1 REQ'D.
1/4-20
1/2
3/8 DRILL
1.166
1/2
.750
.740
D.
1/4
1/2
1 9/16
3/8
DIA. PIT -12
NO. TEETH -12
3/4 REAM
3/8 DRILL
1/4
1/4
9/16
1/4
7/8
1/2
5/16 R.
1/8 FLAT
5/8
-7 BACK PAD - 1 REQ'D.
1/4-20
SMYRNA HIGH SCHOOL
VOCATIONAL MACHINE SHOP
NAME OF PART ARBOR PRESS
PROJECT NO. QTY. REQ'D 1
MAT'L 1018 H.T. S.F SCALE 1=1
TOLERANCE UNLESS OTHERWISE SPECIFIED
DEC ±.002 FRAC ±1/64 ANG ±30'
DRAFTSMAN Rick Robinson DATE 1-24-71



SMYRNA HIGH SCHOOL	
VOCATIONAL MACHINE SHOP	
NAME OF PART ARBOR PRESS	
PROJECT NO.	QTY. REQD. 1
MAT'L 1018 H.T. S.F	SCALE 1:2
TOLERANCE UNLESS OTHERWISE SPECIFIE	
DEC ±.002 FRAC ± 1/64	ANG ± 30'
DRAFTSMAN Rick R.	DATE 2-25-71